Sheet Metal Designer's Handbook

DOCUMENT IDENTIFIER: A-MN-ELEN725-01-0000 Rev B, 23-Nov-1992

ABSTRACT: This handbook summarizes sheet metal design rules.

APPLICABILITY: This handbook is recommended for all mechanical and manufacturing engineers throughout the corporation.

STATUS: APPROVED 23-Nov-1992; use VTX SMC for current status.

This document is confidential and proprietary, and is the property of Digital Equipment Corporation. This is an unpublished work protected under applicable copyright laws.

© Digital Equipment Corporation 1990, 1992. All rights reserved.

Sheet Metal Designer's Handbook

DOCUMENT IDENTIFIER: A-MN-ELEN725-01-0000 Rev B, 23-Nov-1992

REVISION HISTORY:	Rev A,	17-Dec-1990	
	Rev B,	23-Nov-1992	ECO Number NRO01

Document Management Category:	Raw Materials/Mechanical Technology (HPM)
Responsible Department:	Manufacturing Operations
Responsible Person:	Bill Morrill
SMC Writer:	Bob Laslocky

APPROVAL: This document has been approved for use within Digital Equipment Corporation.

William A. Morrill, Manager, Design for Manufacturing,
E/M Design and Support

Direct requests for further information to:

Lisa Modzelewski - Mechanical Manufacturing Engineering /New Products
Use $ VTX ELF for the latest location information.

Use VTX SMC to order copies of this document from Standards and Methods Control. Send distribution questions to JOKUR::SMC or call DTN: 234-4423.

The DIGITAL logo is a trademark of Digital Equipment Corporation.

CONTENTS

Chapter 1 INTRODUCTION 1
1.1 PURPOSE 2
1.2 SCOPE 2
1.3 MECHANICAL MANUFACTURING ENGINEERING/NEW PRODUCTS 2
1.4 HANDBOOK HIERARCHY 2
1.4.1 Rule Format 3
1.4.2 Defining the Rule 3
1.4.3 Architecture 3

Chapter 2 FEATURES 7
2.1 HOLES 8
2.1.1 PIERCING 8
2.1.2 DIAMETERS 10
2.1.3 SLOT 12
2.1.4 HOLES TO FORM 14
2.1.5 HOLE TO BEND 16
2.1.6 SLOT TO BEND 18
2.1.7 HOLE TO HOLE 20
2.1.8 HOLE TO EDGE 22
2.1.9 HOLE THROUGH A BEND 26
2.2 FORMS 28
2.2.1 BENDING 28
2.2.2 RADIUS 30
2.2.3 BEND TO EDGE 34
2.2.4 BEND TO HOLE 36
2.2.5 BEND RELIEF 38
2.2.6 BEND TO BEND 40
2.2.6.1 OFFSET BENDS 42
2.2.7 DIMPLE 44
2.2.8 DIMPLE TO HOLE 46
2.2.9 DIMPLE TO EDGE 48
2.2.10 DIMPLE TO BEND 50
2.2.11 DIMPLE TO DIMPLE 52
2.2.12 GUSSET 54
2.2.13 GUSSET TO HOLE 56
2.2.14 GUSSET TO GUSSET 58
2.2.15 GUSSET TO EDGE 60
2.2.16 GUSSET TO BEND 62
2.2.17 CURL 64
2.2.18 CURL TO HOLE 66

2.2.19 CURL TO BEND 68
2.2.20 CURL TO CURL 70
2.2.21 HEM 72
2.2.22 HEM - TEARDROP 74
2.2.23 HEM - OPEN 76
2.2.24 HEM - CLOSED 78
2.2.25 HEM TO HOLE 80
2.2.26 HEM TO EDGE 82
2.2.27 HEM TO BEND 84
2.2.28 HEM TO HEM 86
2.2.29 RIB 88
2.2.30 RIB TO HOLE 90
2.2.31 RIB TO EDGE 92
2.2.32 RIB TO BEND 94
2.2.33 RIB TO RIB 96
2.2.34 EMBOSSMENT 98
2.2.35 EMBOSSMENT SHAPES 100

2.3 EDGE 102
2.3.1 NOTCH 104
2.3.2 NOTCH TO HOLE 106
2.3.3 NOTCH TO BEND 108
2.3.4 NOTCH THROUGH A BEND 110
2.3.5 NOTCH TO NOTCH 112
2.3.6 TAB 114
2.3.7 TAB TO TAB 116
2.3.8 CORNER CONDITIONS 118
2.3.9 CORNER CONDITION - RADIUS 120
2.3.10 CORNER CONDITION - CHAMFER 122

2.4 SPECIAL FEATURES 124
2.4.1 EXTRUDED HOLE 124
2.4.2 EXTRUDED HOLE - ROLL FORM THREADING 126
2.4.3 EXTRUDED HOLE TO EXTRUDED HOLE 128
2.4.4 EXTRUDED HOLE TO EDGE 130
2.4.5 EXTRUDED HOLE TO BEND 132
2.4.6 LANCE - OPEN 134
2.4.7 LANCE - CLOSED 136
2.4.8 LANCE TO EDGE 138
2.4.9 LANCE TO BEND 140
2.4.10 LANCE TO HOLE 142
2.4.11 LANCE TO LANCE 144
2.4.12 LOUVERS 146
2.4.13 COUNTERSINK 148
2.4.14 COUNTERSINK - FORMED 150
2.4.15 COUNTERSINK - COINED 152
2.4.16 COUNTERSINK TO COUNTERSINK 154

2.4.17 COUNTERSINK TO EDGE 156
2.4.18 COUNTERSINK TO BEND 158
2.4.19 COUNTERBORE 160
2.4.20 COUNTERBORE - COINED 162
2.4.21 COUNTERBORE TO COUNTERBORE 164
2.4.22 COUNTERBORE TO EDGE 166
2.4.23 COUNTERBORE TO BEND 168
2.4.24 SEMI-PIERCED HOLE 170
2.4.25 SEMI-PIERCED HOLE TO FORM 172
2.4.26 SEMI-PIERCED HOLE TO BEND 174
2.4.27 SEMI-PIERCED HOLE TO SEMI-PIERCED HOLE 176
2.4.28 SEMI-PIERCED HOLE TO EDGE 178

2.5 FINISHING 179

2.6 JOINING 180
2.6.1 Slot and Tab 180
2.6.2 RIVETS 182
2.6.3 RIVETS - BLIND 184
2.6.4 WELDING 186
2.6.5 INSERTS 188
2.6.6 AKH (RIVET, FASTENER) 190

GLOSSARY 193

Appendix A REFERENCED DOCUMENTS 195

A.1 Digital Documents, EL-Class 195

A.2 Documents External to Digital 195

A.3 Ordering Information 195

INDEX

FIGURES

1 Metals Architecture 4
2 Holes, (Piercing Hole and Piercing Clearance) 9
3 Diameters (Round Hole and Slot) 11
4 Slot (Round Hole and Slot) 13
5 Holes to Form 15
6 Hole to Bend 17
7 Slot to Bend 19
8 Hole to Hole (Hole to Hole/Edge) 21
9 Hole to Edge (Hole to Hole/Edge) 23
10 Hole to Edge (Bulge Condition) 25
11 Hole Through a Bend (Hole Through Bends) 27
12 Bends 29

13	Radius	33
14	Bend to Edge	35
15	Bend to Hole	37
16	Bend Reliefs	39
17	Offset Bends	43
18	Dimple	45
19	Dimple to Hole	47
20	Dimple to Edge	49
21	Dimple to Bend	51
22	Dimple to Dimple	53
23	Gusset	55
24	Gusset to Hole	57
25	Gusset to Gusset	59
26	Gusset to Edge	61
27	Curl	65
28	Curl to Hole	67
29	Curl to Bend	69
30	Hem	73
31	Hem - Teardrop	75
32	Hem - Open	77
33	Hem - Closed	79
34	Hem to Hole	81
35	Hem to Bend	85
36	Rib	89
37	Rib to Hole	91
38	Rib to Edge	93
39	Rib to Bend	95
40	Rib to Rib	97
41	Embossment	99
42	Embossment Shapes	101
43	Notch	105
44	Notch to Hole	107
45	Notch to Bend	109
46	Notch Through a Bend	111
47	Notch to Notch	113
48	Tab	115
49	Tab to Tab	117
50	Corner Conditions	119
51	Corner Condition - Radius	121
52	Corner Condition - Chamfer	123
53	Special Features (Extruded Hole)	125
54	Extruded Hole - Roll Form Threading	127
55	Extruded Hole to Extruded Hole	129
56	Extruded Hole to Edge	131
57	Extruded Hole to Bend	133
58	Lance - Open	135

59	Lance - Closed	137
60	Lance to Bend	141
61	Lance to Hole	143
62	Louvers	147
63	Countersink	149
64	Countersink - Formed	151
65	Countersink - Coined	153
66	Countersink to Countersink	155
67	Countersink to Edge	157
68	Countersink to Bend	159
69	Counterbore	161
70	Counterbore - Coined	163
71	Counterbore to Counterbore	165
72	Counterbore to Edge	167
73	Counterbore to Bend	169
74	Semi-Pierced Hole	171
75	Semi-Pierced Hole to Form	173
76	Semi-Pierced Hole to Bend	175
77	Semi-Pierced Hole to Semi-Pierced Hole	177
78	Slot and Tab	181
79	Rivets	183
80	Rivets - Blind	185
81	Standard Welding Symbols	187
82	AKH (Rivet, Fastener)	191

CHAPTER 1

INTRODUCTION

1.1 PURPOSE

The *Sheet Metal Designer's Handbook* summarizes sheet metal design rules used to optimize the manufacturing of sheet metal parts and enclosures. By designing sheet metal parts and enclosures that comply to this handbook, the engineer is assured that the product will be manufacturable:

- Using standard manufacturing processes
- Using rapid prototyping methodologies
- Within Digital's internal sheet metal facilities and or approved external vendor base
- Permitting the shortest possible volume manufacturing start-up
- Providing designed-in quality that will ensure high production yields and reduced rework costs
- Leading to Six-Sigma compliance which requires that the specification width be twice the process capability

1.2 SCOPE

This handbook applies to sheet metal having a thickness up to 6 mm = (0.25 inch). This handbook is intended to be an overall knowledge base; where more detail is required, see a mechanical Producibility Engineer.

1.3 MECHANICAL MANUFACTURING ENGINEERING/NEW PRODUCTS

The Mechanical Manufacturing Engineering/New Products core team is responsible for the contents of the handbook, and the members of this team must review and approve all proposed changes. Send all comments and proposed changes to Lisa Modzelewski, WFO/C4, DTN 242-2471, WFOOFF::MODZELEWSKI.

Core Team members include:

Dave Albani	NPOVAX::ALBANI
Ron Colby	WFOOFF::COLBY_R
Jim Fay	MILVAX::FAY
Ray Gladu	EMDS::GLADU
Larry Mazzone	PROXY::MAZZONE
Lisa Modzelewski	WFOOFF::MODZELEWSKI.
Bill Morrill	EMDS::MORRILL

1.4 HANDBOOK HIERARCHY

```
Technology (volume)--------------Metals
Process (book)-------------------Sheet metal
Specific Information (chapter)---Design features and considerations
Feature class (section)----------Holes, bends, and so on
Design rule (specific rule)------Hole to bend, and so on
```

1.4.1 Rule Format

The following format has been used in the creation of the Sheet Metal Design rules.

```
-----------------------------------------------------------------
SECTION: Basic process classification to which the rules
         apply (example, Sheet Metal)
=================================================================
OPERATION: Type operation to which the rule applies
   (example, Stamping Process - Holes)
-----------------------------------------------------------------
RULE DESCRIPTION: General description of the rule

RULE CATEGORY: Axiom, Principle, Guideline or Information
       (Rules are used to describe how and why design
        choices are and should be made)
-----------------------------------------------------------------
REASON: The underlying fact or cause that provides the
        logic for the rule
-----------------------------------------------------------------
ALTERNATIVE: Provides a design choice outside the limits of
             the (recommended) rule
-----------------------------------------------------------------
ALTERNATIVE IMPACT: The effect(s) that will be incurred by
     not using or following the prescribed rule
```

1.4.2 Defining the Rule

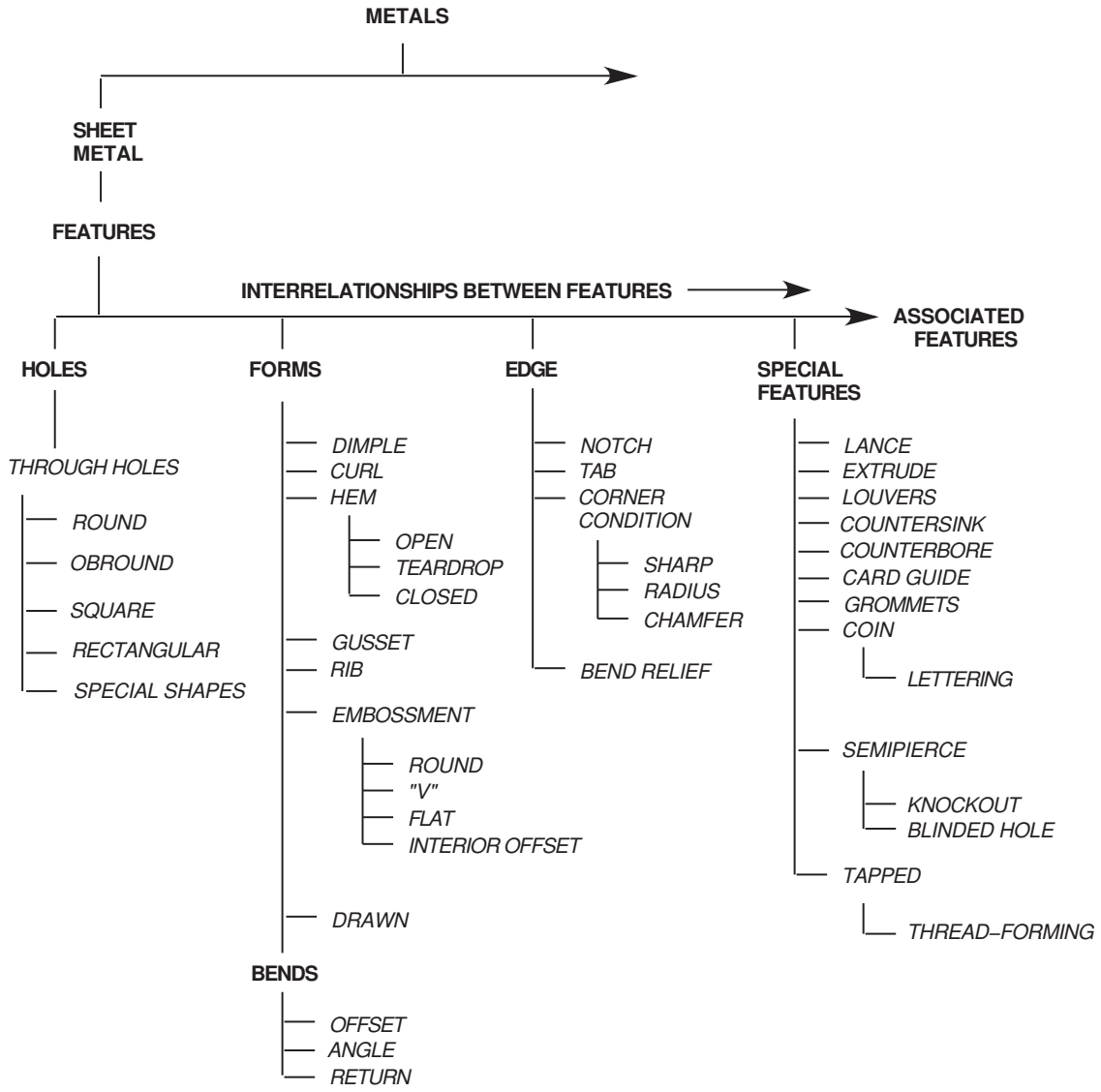
The Sheet Metal Design Rules are used to describe how or why design choices are, or should be, made. The following catagories describe rules; each rule begins with the corresponding banner.

PRINCIPLE	Basic knowledge that is generally applicable
GUIDELINE	A recommendation that applies to a specific situation or configuration
INFORMATION	Definition or knowledge that aids in understanding

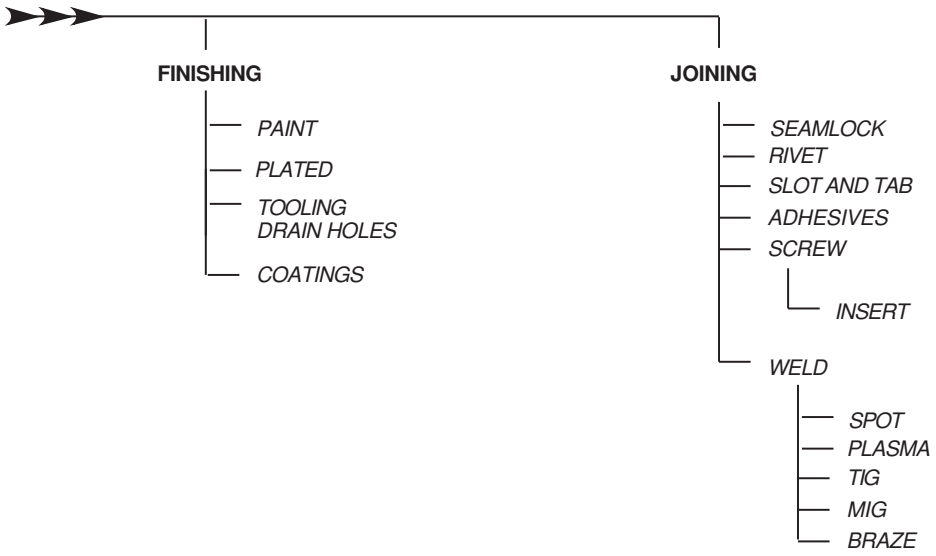
1.4.3 Architecture

The following is the metals architecture that is used to create the Sheet Metal Design Handbook.

Figure 1: Metals Architecture



A S S O C I A T E D F E A T U R E S



CHAPTER 2

FEATURES

2.1 HOLES

2.1.1 PIERCING

INFORMATION

RULES:

1. The most economical method of producing holes in sheet metal is to use the process known as piercing or punching.
2. The following Charalteristics are seen from the direction of the punch: rollover, a burnished straight section, a fractured angular section, and a burr. See Figure 2, Holes, (Piercing Hole and Piercing Clearance), A.

REASONS: There are two basic standard manufacturing processes available for piercing holes in sheet metal. They are:

- Hard dies (complex high-volume, simple low-volume)
- Numerically controlled punch press (low- and medium-volume).

The relative physical condition of a pierced hole is a function of the punch and die clearance. See Figure 2, Holes, (Piercing Hole and Piercing Clearance), B.

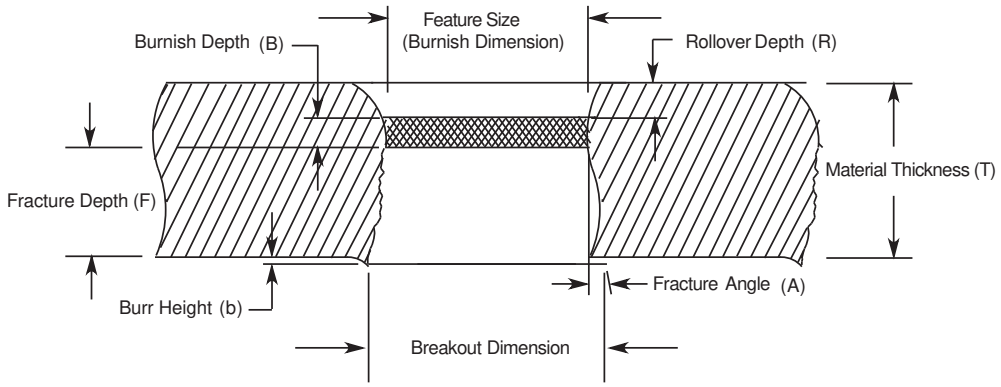
ALTERNATIVES: There are usually other more expensive methods to produce holes in sheet metal without a fractured edge. Examples include: Hole-shaving dies, drilling, laser cutting, chemical etching, and electrical-discharge machining (EDM).

ALTERNATIVE IMPACT: These other methods of producing holes require special equipment that cannot produce adequate volumes cost-effectively.

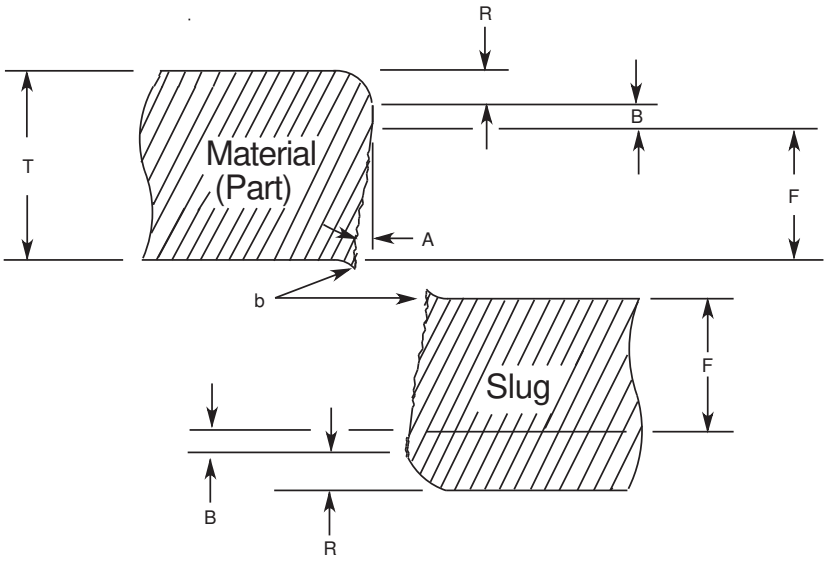
Special hard dies, such as hole-shaving dies may require additional, more costly processes.

Figure 2: Holes, (Piercing Hole and Piercing Clearance)

A. Piercing Hole



B. Piercing, with clearance of 8 to 10% of material thickness



T = Material Thickness
A = Fracture Angle ---------- 7° TO 11°
R = Rollover ---------- 6% TO 8% T
B = Burnish ---------- 25% TO 40% T
F = Fracture ---------- 50% TO 60% T
b = Burr ---------- Normal

2.1.2 DIAMETERS

PRINCIPLE

RULE:

The minimum diameter of a hole (or minor dimension of a slot) should be equal to the material thickness or 1.00 mm (0.04 inch), whichever is greater. See Figure 3, Diameters (Round Hole and Slot).

REASON: Punching a hole that is less than the material thickness or less than 1.00 mm (0.04 inch) usually requires special tooling that leads to increased tool maintenance and decreased tool life.

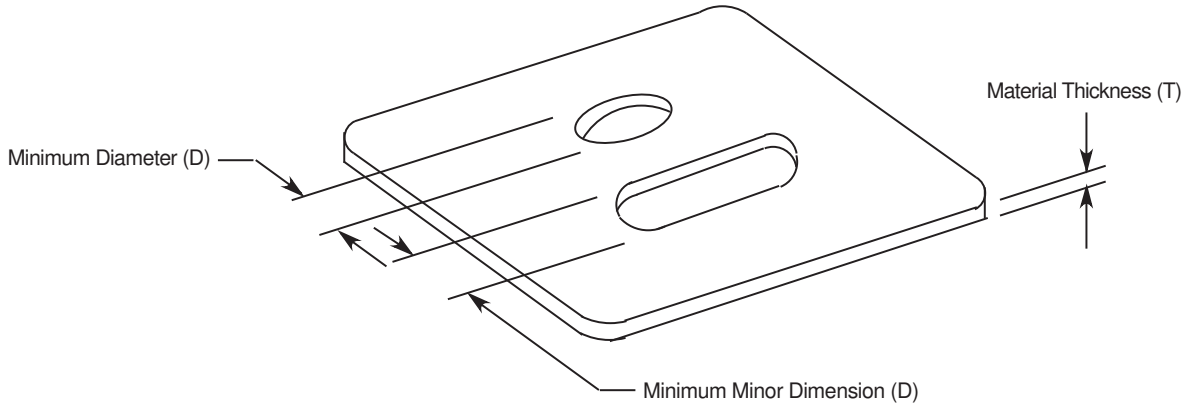
ALTERNATIVES:

There are other methods of producing holes with (minor) diameters of less than one stock thickness, or a diameter of less than 1.00 mm (0.04 inch). Examples include, but are not limited to:

- Specially designed tooling/dies
- Milling/drilling
- Electrical-discharge machining (EDM)

ALTERNATIVE IMPACT: These other methods of producing holes may require special tooling or secondary operations that lead to increased process and tooling costs.

Figure 3: Diameters (Round Hole and Slot)



Rule: D (min) = T or 1.00 mm (0.04 in), whichever is larger.

2.1.3 SLOT

GUIDELINE

RULE:

The minimum width of a slot is equal to the material thickness or 1.00 mm (0.04 inch), whichever is greater. There are no constraints on the length of a slot.

REASON: Punching a slot that is less than the material thickness or 1.00 mm (0.04 inch) wide usually requires special tooling and/or processes that lead to increased tool maintenance and decreased tool life.

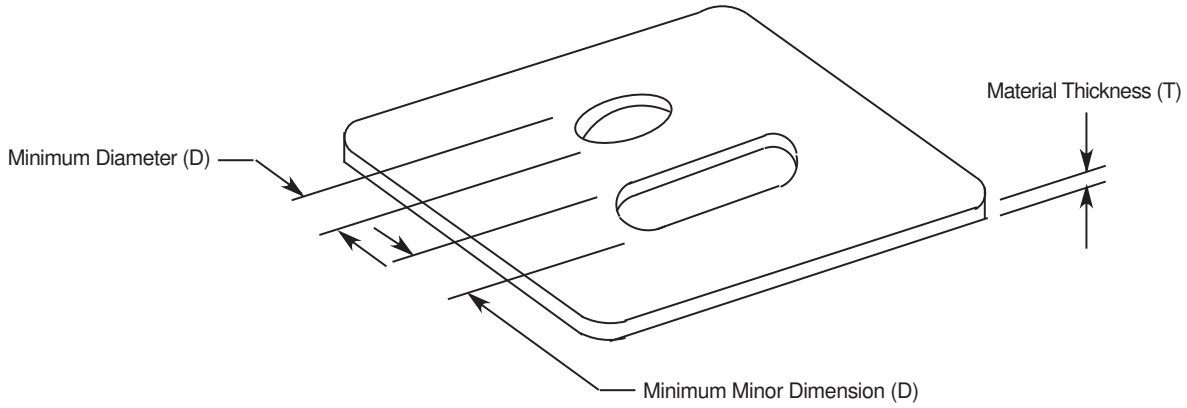
See a Producibility Engineer for recommendations on maximum slot length, based upon specific design requirements and manufacturing constraints.

ALTERNATIVES: There are other methods of producing slots narrower than the rule suggests. Examples include, but are not limited to:

- Specially designed tooling/dies
- Laser/milling

ALTERNATIVE IMPACT: These other methods of producing slots may require special tooling or secondary operations leading to increased process and tooling costs.

Figure 4: Slot (Round Hole and Slot)



Rule: D (min) = T or 1.00 mm (0.04 in), whichever is larger.

2.1.4 HOLES TO FORM

GUIDELINE

RULE:

The minimum recommended distance that the edge of a hole should be from a form is three times the material thickness plus the form radius. See Figure 5, Holes to Form.

REASONS: Holes positioned too close to a formed feature may result in hole distortion. See Figure 5, Holes to Form.

Distortion can lead to manufacturability and/or quality problems.

ALTERNATIVES: Holes may be located much closer to a formed feature than the rule suggests through the use of other manufacturing methods (for example, drilling or using secondary operations, such as to punch after form).

Holes may also be located much closer to a formed feature than the rule suggests by using relief cuts/slots.

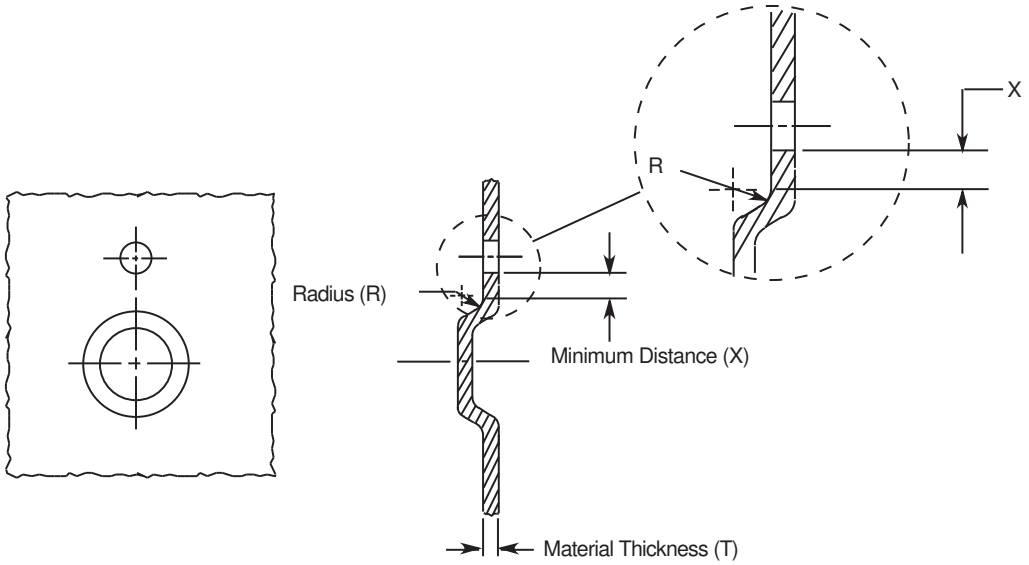
If the design is not impaired by hole distortion, you may also add a note to the drawing/database stating that hole distortion is allowable.

ALTERNATIVE IMPACT: These other methods of producing holes require special tooling and/or secondary operations leading to increased process and/or tooling costs.

Design considerations and/or constraints may impair the addition of form reliefs. They also make it necessary to incur operational and/or tooling cost increases.

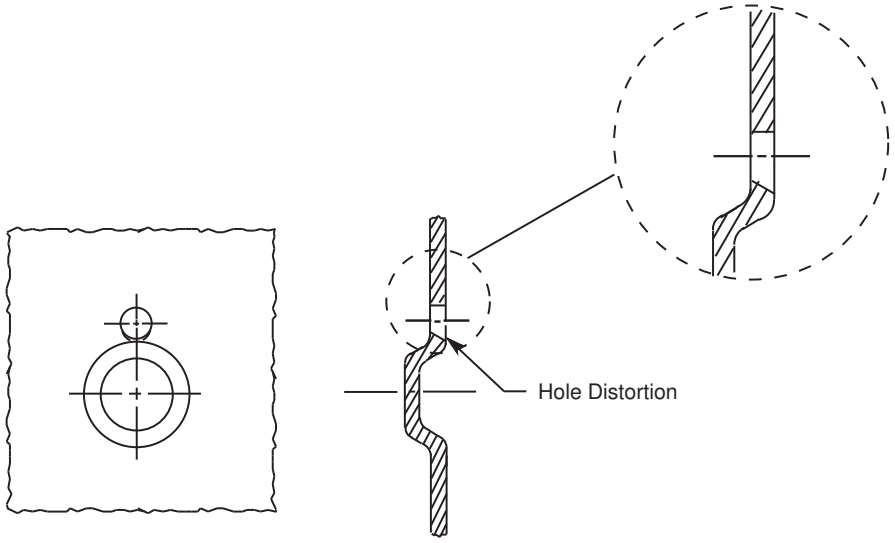
Figure 5: Holes to Form

A. Hole To A Form (Embossment)



Rule: X (Minimum) = 3T+R

B. Hole Too Close To A Form (Embossment)



2.1.5 HOLE TO BEND

PRINCIPLE

RULE:

The minimum recommended distance that the edge of a hole should be from a bend is two times the material thickness plus the bend radius. See Figure 6, Hole to Bend.

REASONS: Holes positioned too close to a bend may result in hole distortion. See Figure 6, Hole to Bend.

Distortion can lead to manufacturability and/or quality problems.

ALTERNATIVES: Holes may be located much closer to a bend than the rules suggests through the use of other manufacturing methods (for example, drilling or using secondary operations, such as to punch after bending).

Holes may also be located much closer to a bend than the rule suggests through the use of relief cuts/slots. See Figure 6, Hole to Bend.

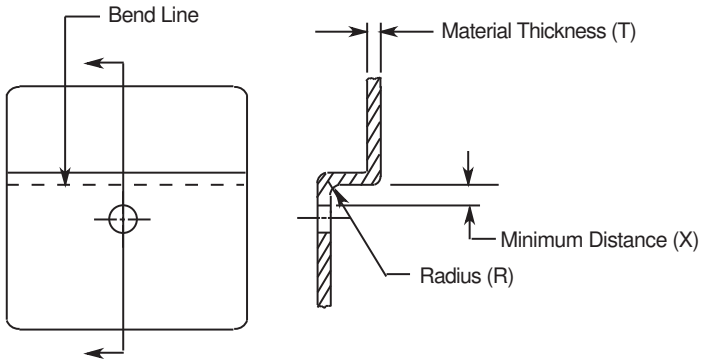
If the design is not impaired by hole distortion, a note can be added to the drawing/database stating that hole distortion is allowable (be specific on location).

ALTERNATIVE IMPACT: These other methods of producing holes require special tooling and/or secondary operations leading to increased process and tooling costs.

Design considerations and/or constraints may impair the addition of bend reliefs. They also make it necessary to incur operational and/or tooling cost increases.

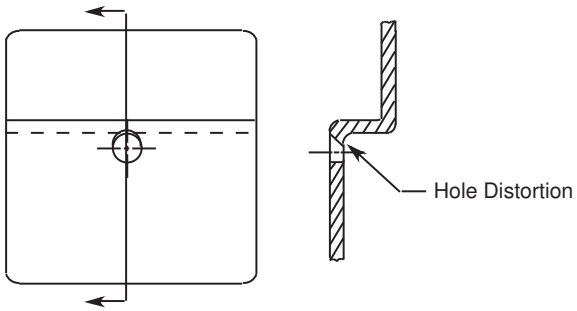
Figure 6: Hole to Bend

A. Bend To Hole

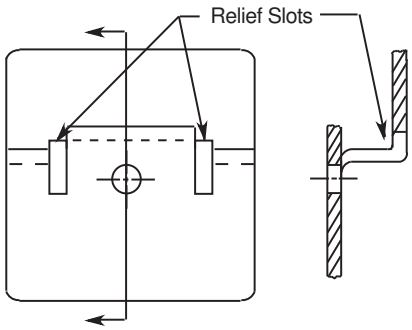


Rule: X (minimum) = 2 T + R

B. Hole Distortion



C. Relief Slots



2.1.6 SLOT TO BEND

GUIDELINE

RULE:

The minimum distance from the inside surface of a bend to the edge of a slot is directly proportional to the length of the slot, material thickness, and the radius of the bend. See Figure 7, Slot to Bend.

REASONS: Bending closer to a slot than the rule suggests may lead to manufacturability (tooling and process) or quality problems (bent flange squareness, or slot distortion, or both).

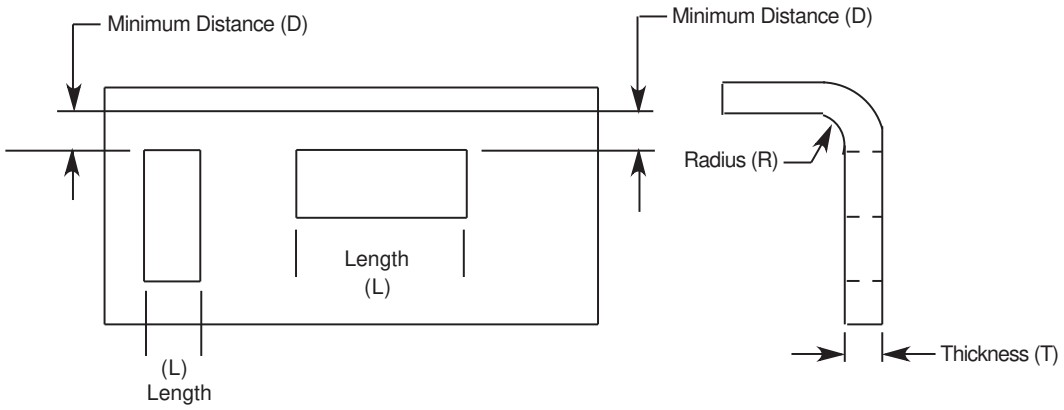
ALTERNATIVES: Slot (hole) through a bend.

There are other methods of producing a bent flange closer to a slot than the rule suggests. Examples include, but are not limited to:

- Specially designed tooling/dies
- Special processes (for example, punch slot after bending)

ALTERNATIVE IMPACT: These other methods of producing a bent flange close to a slot may require special tooling or secondary operations that lead to increased process and tooling costs.

Figure 7: Slot to Bend



Rule:

When L is less than 25.4 mm (1.0 in), then D (minimum) = 2T + R
When L is 25.4 mm (1.0 in) to 50.8 mm (2.0 in), then D (minimum) = 2.5 T + R
When L is greater than 50.8 mm (2.0 in), then D (minimum) = 3T + R

2.1.7 HOLE TO HOLE

PRINCIPLE

RULE:

The minimum recommended distance between holes is directly proportional to the size and shape of the hole feature and the material thickness. See Figure 8, Hole to Hole (Hole to Hole/Edge).

REASONS: Holes punched too close together may result in distortion of the hole and/or the web area between them.

Numerous holes punched too close together may result in the distortion or warpage (oil-canning) of the material.

ALTERNATIVES: Combine the holes into a single (larger) hole. Examples include, but are not limited to:

- Two round holes into one slot
- Two slots into one larger cutout

Combine the holes into a single (larger) hole and then attach a separate, specially perforated insert (see subhead 2.6 Joining).

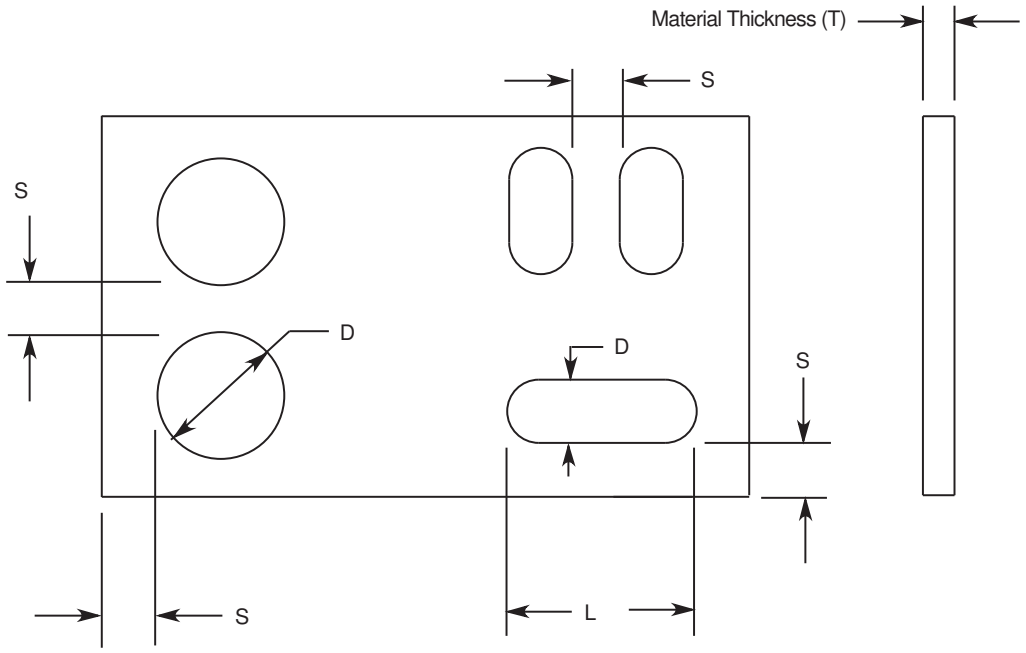
There are other methods of producing holes closer than the rule suggests. Examples include, but are not limited to:

- Specially designed tooling/dies
- Milling
- Electrical-discharge machining (EDM)

ALTERNATIVE IMPACT: Positive or negative impacts of the alternatives would have to be evaluated based upon specific design requirements. See a Producibility Engineer.

These other methods of producing holes require special tooling and/or secondary operations that lead to increased process and tooling costs.

Figure 8: Hole to Hole (Hole to Hole/Edge)



D = Diameter or Minimum Dimension of Slot
L = Length
T = Material Thickness
S = Minimum Dimension

When D is 1 T to 5 T then minimum S = 1.5 T
When D is 5 T to 10 T then minimum S = 2.0 T
When D is greater than 10 T then minimum S = 3.5 T
When L is 1 T to 10 T then minimum S = 2 T
When L is 10 T to 25 T then minimum S = 4 T
When L is greater than 25 T then minimum S = 6 T
See Producibility Engineer for specific recommendation

2.1.8 HOLE TO EDGE

PRINCIPLE

RULE:

A minimum distance between a hole and the edge of the material must be maintained. The minimum recommended distance between the hole and the edge is directly proportional to the size and shape of the hole and the material thickness. See Figure 9, Hole to Edge (Hole to Hole/Edge).

REASON: Holes punched too close to an edge may result in distortion of the hole and/or the web area between the hole and the edge, creating a bulge. See Figure 10, Hole to Edge (Bulge Condition).

ALTERNATIVES: Blend/join (combine) the hole with the edge. For example, make a round hole near the edge into a slot. See Figure 10, Hole to Edge (Bulge Condition).

Extend the edge (local to the hole) to satisfy the rule. For example, add a tab. See Figure 10, Hole to Edge (Bulge Condition).

There are other methods of producing holes closer to an edge than the rule suggests. Examples include, but are not limited to, specially designed tooling/dies, milling, and laser.

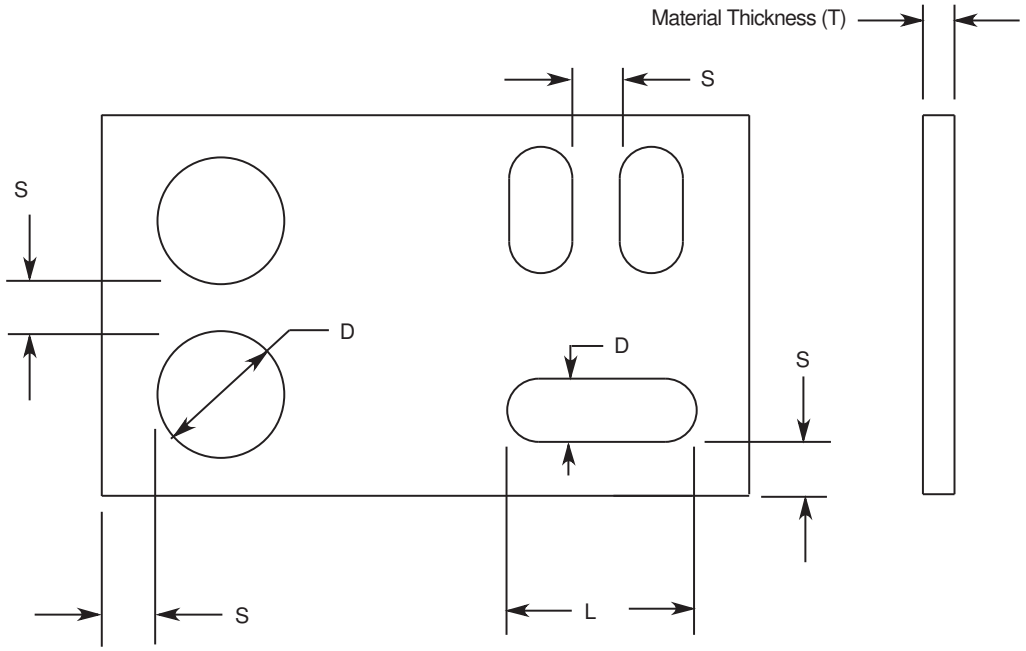
If the design is not impaired by the edge distortion (bulge), a note could be added to the drawing/database stating that edge distortion is allowable (be specific on location).

ALTERNATIVE IMPACT: Positive or negative impacts of the alternatives have to be evaluated based upon specific design requirements. See a Producibility Engineer.

These other methods of producing holes require special tooling and/or secondary operations that lead to increased process and tooling costs.

Distortion could lead to manufacturing and/or quality problems.

Figure 9: Hole to Edge (Hole to Hole/Edge)

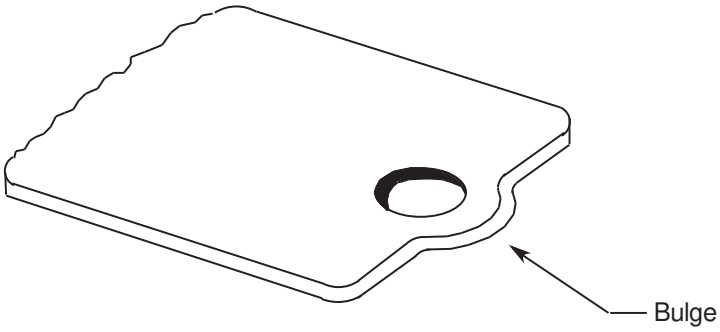


D = Diameter or Minimum Dimension of Slot
L = Length
T = Material Thickness
S = Minimum Dimension

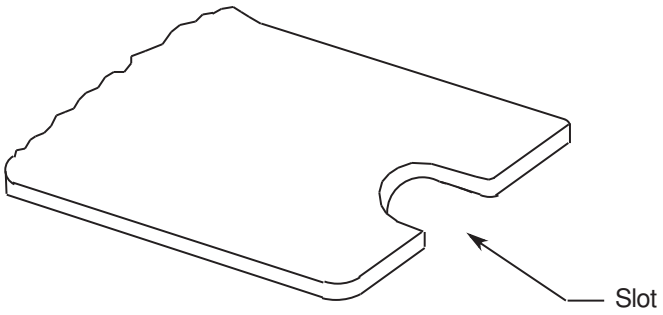
When D is 1 T to 5 T then minimum S = 1.5 T
When D is 5 T to 10 T then minimum S = 2.0 T
When D is greater than 10 T then minimum S = 3.5 T
When L is 1 T to 10 T then minimum S = 2 T
When L is 10 T to 25 T then minimum S = 4 T
When L is greater than 25 T then minimum S = 6 T
See Producibility Engineer for specific recommendations.

Figure 10: Hole to Edge (Bulge Condition)

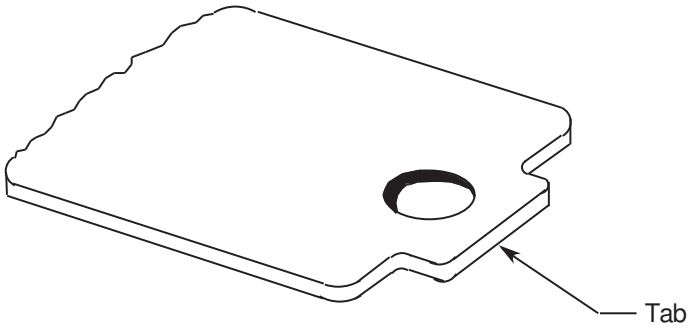
A. Bulge Condition



B. Make a Slot



C. Add a Tab



2.1.9 HOLE THROUGH A BEND

GUIDELINE

RULE:

When requiring a hole through a bend, the leading edges of the hole should extend a minimum distance beyond both sides of the bend, equal to the thickness of material plus the bend radius or two times the material thickness, whichever is greater. See Figure 11, Hole Through a Bend (Hole Through Bends).

REASONS: A hole, internal to a bend, that does not extend beyond the bend will distort during the bending operation (see rule).

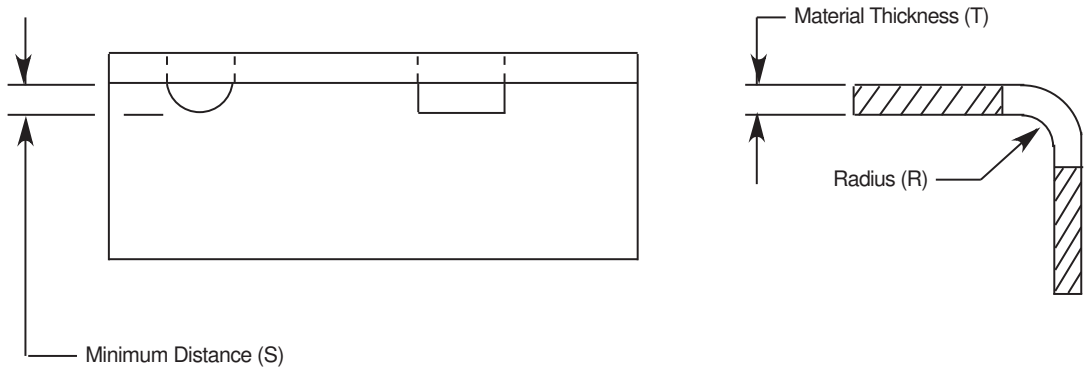
Distortion could lead to manufacturing or quality problems.

ALTERNATIVES: There are other methods of producing holes internal to bends without distortion. Examples include, but are not limited to: specially designed tooling/dies, machining after bending, milling, laser.

If the design is not impaired by the hole distortion , a note could be added to the drawing/database stating that hole distortion is allowable (be specific on feature and location).

ALTERNATIVE IMPACT: These other methods of producing holes may require special tooling or secondary operations that lead to increased process and tooling costs.

Figure 11: Hole Through a Bend (Hole Through Bends)



Rule: S (minimum) = 2 T or R + T, whichever is larger.

2.2 FORMS

2.2.1 BENDING

INFORMATION

RULES:

1. Bend angles should have a tolerance of plus or minus one-half a degree at a location adjacent to the bend(s). See Figure 12, Bends.
2. Multiple bends on the same plane and bent in the same direction promote ease of manufacturing. See Figure 12, Bends.
3. Large sheet metal parts with small bent flanges should be avoided.

REASONS: The sheet metal bending tolerance (±1/2 degree) is a standard. See *DEC STD 187-0 Mechanical Fabrication Workmanship Standards*.

Due to several factors, such as flatness of material, length of bent flanges verses material thickness, the measuring of bend angles, in any location other than adjacent to the bend(s), is very difficult.

Tooling/die construction and part transfer are greatly simplified if all bends are on the same plane and are formed in the same direction.

Small bent flanges on large sheet metal parts are very difficult to manufacture and can lead to quality issues.

ALTERNATIVES: Bends may be formed at tolerances tighter than those specified in the rule through the use of dedicated tooling/dies.

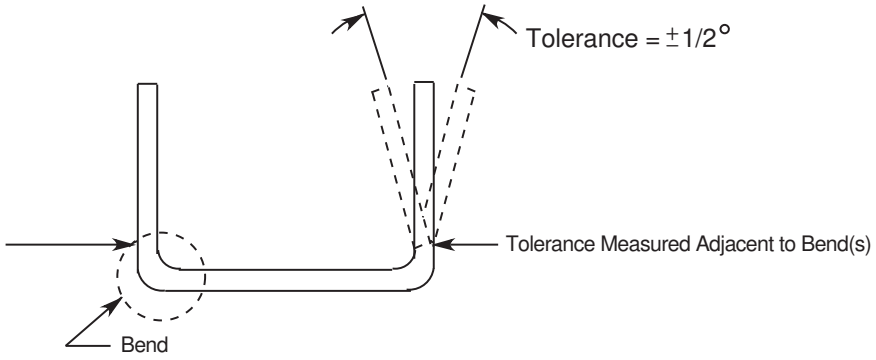
Bends may be formed in alternate directions through the use of special tooling/dies and/or multiple processes.

Small bent flanges on large sheet metal parts may be produced by using multiperson operations and/or special equipment.

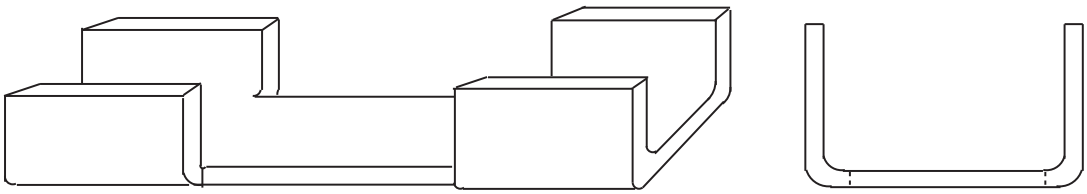
ALTERNATIVE IMPACT: Dedicated/special equipment or multiple processes lead to increased tooling and/or process costs.

Figure 12: Bends

A. Bending Tolerance



B. Ease of Manufacturing



Note:
Flanges on the same plane should be bent in the same direction for ease of manufacturing.

2.2.2 RADIUS

PRINCIPLE

RULES:

1. In low carbon steel sheet metal, the minimum radius of a bend should be one-half the material thickness or 0.80 mm (0.03 inch), whichever is larger. See Figure 13, Radius.
2. In aluminum sheet metal, the minimum radius of a bend should be the material thickness or greater.

NOTE

Due to standard tooling, a 0.80 mm (0.03 inch) or 1.50 mm (0.06 inch) bend radius is preferred.

REASONS: Bending with a radius that is less than one-half the material thickness or less than 0.80 mm (0.03 inch) may cause fracturing of the material surface on the outside of the bend. See Figure 13, Radius.

Bending with a sharp radius (especially in heavy or thick materials) may cause a bulge condition on the edges of the bend. See Figure 13, Radius.

Material fractures are most common in aluminum due to grain. Therefore, a larger radius is preferred.

ALTERNATIVES: There are other methods of producing radii less than half the material thickness or less than 0.80 mm (0.03 inch). Examples include, but are not limited to:

- Specially designed tooling/dies
- Special processes

Bulging of the edges of a bend may be eliminated by using specially designed tooling or additional manufacturing operations. Examples include, but are not limited to:

- Edge constraining forming tools/dies
- Machining (milling) off the bulge after bending

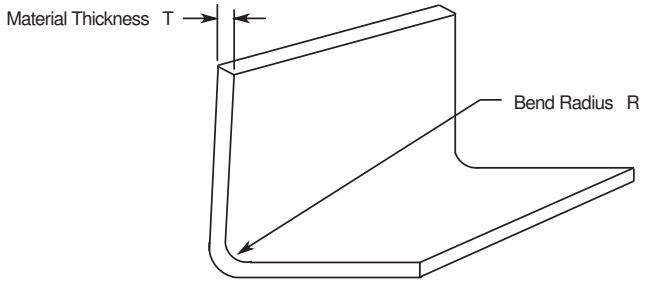
If the design is not impaired by the material cracking or bulging, you could add a note to the drawing/database stating that material cracking deformation is allowable (be specific on location).

ALTERNATIVE IMPACT: These other methods of producing bends without (or by the elimination of) fractures or distortion may require special tooling and/or secondary operations that lead to increased process and tooling costs.

Material fractures and/or distortion could lead to manufacturability and/or quality problems.

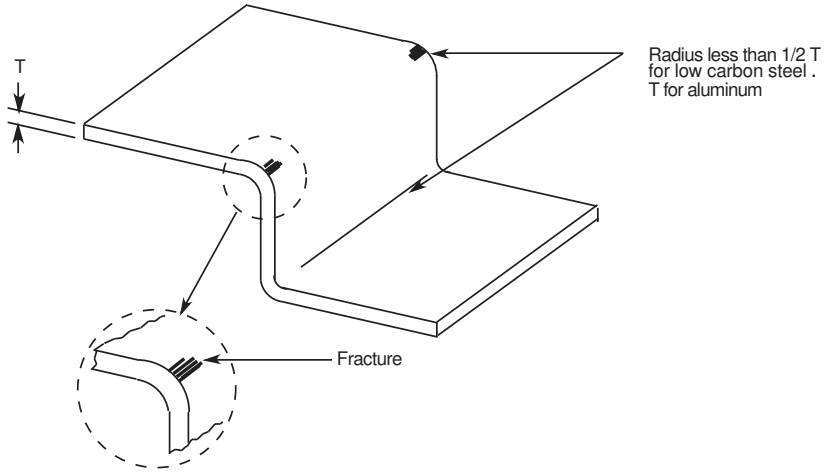
Figure 13: Radius

A. Bend Radius

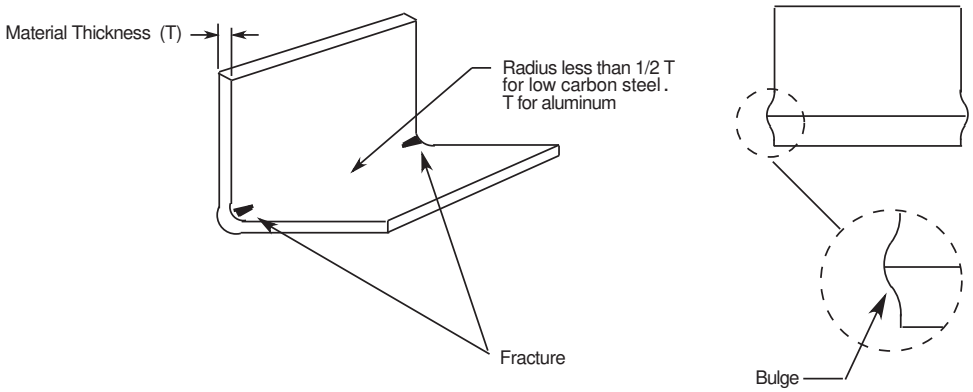


Rule: R (minimum) = 1/2 T or 0.80 mm (0.03 inch) for low carbon steel, whichever is larger.
Rule: R (minimum) = T or 1.50 mm (0.06 inch) for Aluminum, whichever is larger.

B. Bend Fracture



C. Bulging



2.2.3 BEND TO EDGE

GUIDELINE

RULE:

The minimum height of a bent flange (or the minimum distance from the inside surface of the bend to the edge) is directly related to the material thickness, bend radius, and length of bend. See Figure 14, Bend to Edge.

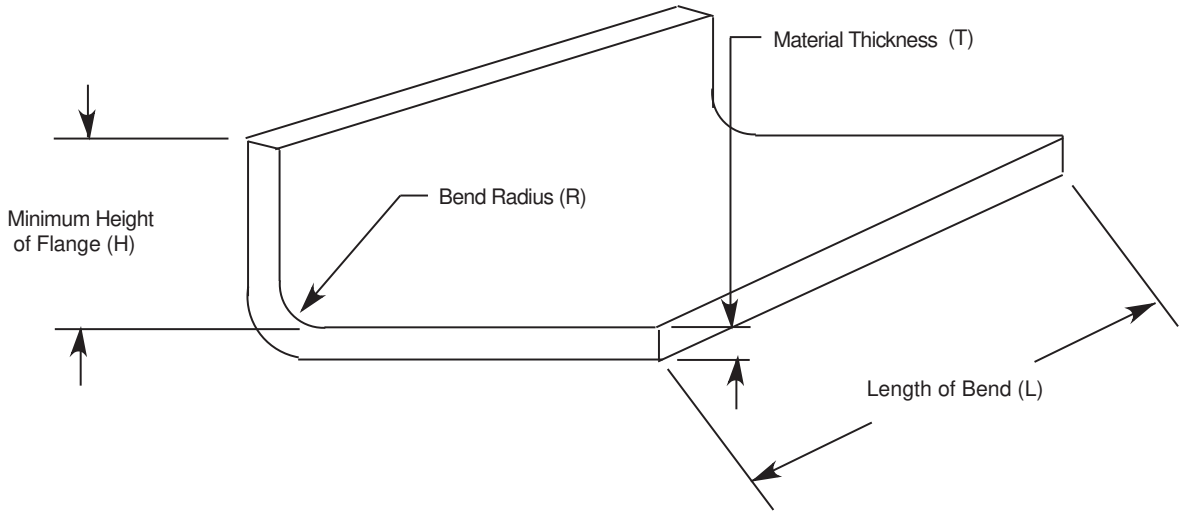
REASON: Bending a flange with a height less than indicated by the rule may lead to manufacturability (tooling and process) and/or quality (flange squareness) problems.

ALTERNATIVES: There are other methods of producing a bent flange with a height less than the rule suggests. Examples include, but are not limited to:

- Specially designed tooling/dies
- Special processes (for example, adding material to flange length and cutting excess material after bending)

ALTERNATIVE IMPACT: These other methods of producing a bent flange may require special tooling or secondary operations that lead to increased process and tooling costs.

Figure 14: Bend to Edge



Rule:

For L up to 304.8 mm (12 in), H (minimum) = 4T + R
For L greater than 304.8 mm (12 in), H (minimum) = 12.7 mm (0.50 in)

2.2.4 BEND TO HOLE

PRINCIPLE

RULE:

The minimum recommended distance that the edge of a hole should be from a bend is two times the material thickness plus the bend radius. See Figure 15, Bend to Hole.

REASONS: Holes positioned too close to a bend may result in hole distortion. See Figure 15, Bend to Hole.

Distortion could lead to manufacturability and/or quality problems.

ALTERNATIVES: Holes may be located much closer to a bend than the rule suggests by using other manufacturing methods, for example, drilling, or using a secondary operation (such as to punch after bending).

Holes may be located much closer to a bend than the rule suggests using bend reliefs. See Figure 15, Bend to Hole.

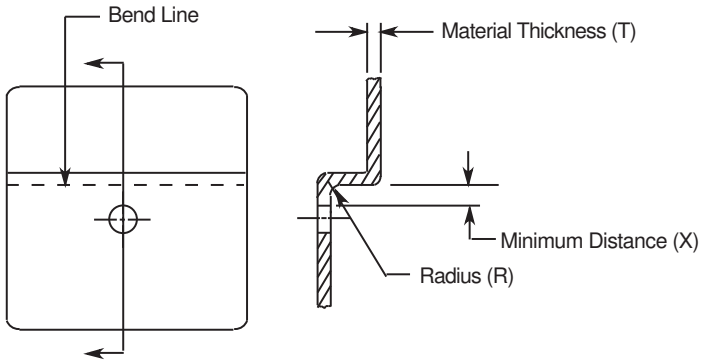
If the design is not impaired by hole distortion, a note could be added to the drawing /database stating that hole distortion is allowable (be specific on location).

ALTERNATIVE IMPACT: These other methods of producing holes require special tooling and/or secondary operations that lead to increased process and/or tooling costs.

Positive or negative impacts of adding bend reliefs would have to be evaluated based upon specific design requirements. See a Producibility Engineer.

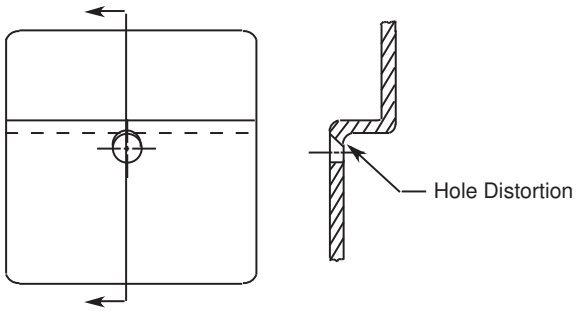
Figure 15: Bend to Hole

A. Bend To Hole

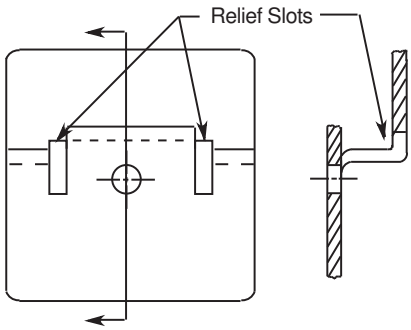


Rule: X (minimum) = 2 T + R

B. Hole Distortion



C. Relief Slots



2.2.5 BEND RELIEF

PRINCIPLE

RULES:

1. When bends do not extend the entire length of a sheet metal surface (creating a flange), the bend relief depth is equal to a minimum of one material thickness plus the bend radius.
2. The minimum width of a bend relief is one material thickness or 1.50 mm (0.06 inch), whichever is greater.

   See Figure 16, Bend Reliefs.

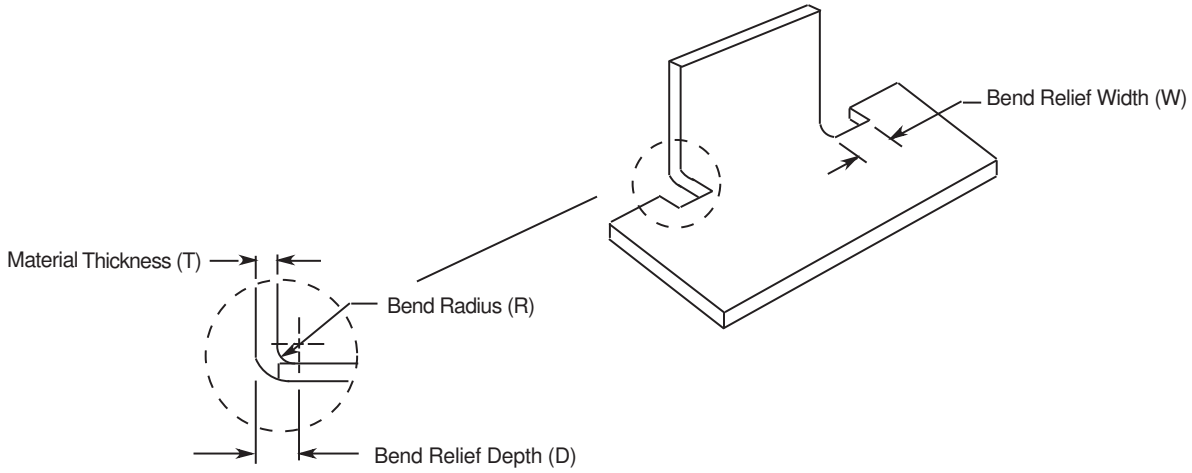
REASON: Forming bends without reliefs as stated in the rule causes the material to tear or crack at the point(s) where the bend blends into the surface. See Figure 16, Bend Reliefs.

ALTERNATIVE: If the design is not impaired by the material tearing or cracking, a note can be added to the drawing/data base stating that material tearing or cracking is allowable (be specific on location).

ALTERNATIVE IMPACT: Material tearing or cracking can lead to design, manufacturability, and quality problems.

Figure 16: Bend Reliefs

A. Bend Reliefs

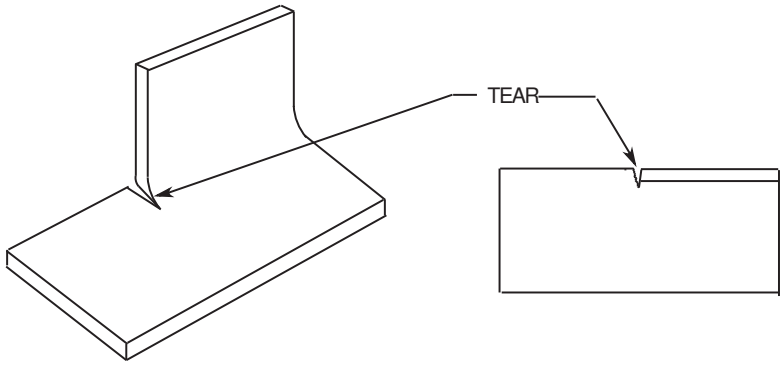


Rules:

W (minimum) = T or 1.50 mm (0.06 in), whichever is larger.

D (minimum) = T + R

B. No Bend Relief



2.2.6 BEND TO BEND

GUIDELINE

RULE

See a Producibility Engineer for recommendations specific to design requirements.

REASON: There are too many variations to specify a general rule for minimum distance between bends.

ALTERNATIVES: N/A

ALTERNATIVE IMPACT: N/A

2.2.6.1 OFFSET BENDS

GUIDELINE

RULE:

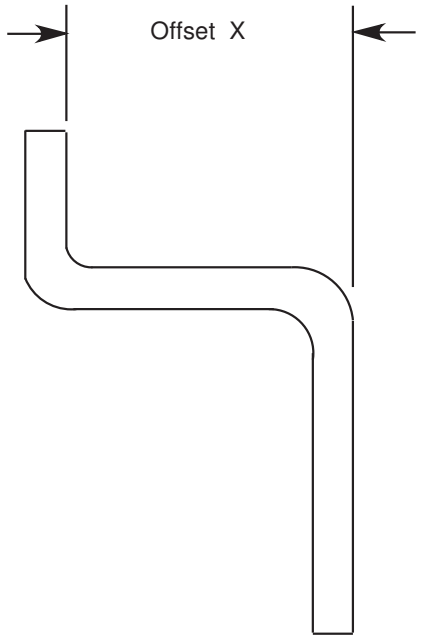
Offset bends should be specified using standard units of measure, whether metric or English (example, all metric offsets should be in increments of 1.50 mm; in English, all offsets should be in increments of 0.06 inch). See Figure 17, Offset Bends.

REASON: When specifying offset bends based upon the rule, standard manufacturing processes and tooling/dies may be used. For more information about available tooling, see a Producibility Engineer.

ALTERNATIVE: Offset bends may be designed/specified using *any* unit of measure.

ALTERNATIVE IMPACT: Nonstandard units of measure require special tooling/dies that lead to increased tooling costs. See a Producibility Engineer.

Figure 17: Offset Bends



Rule:

X (Offset dimension) = standard units of measure

Examples: Metric = increments of 1.5 millimeters
English = increments of 0.06 inch

2.2.7 DIMPLE

GUIDELINE

RULE:

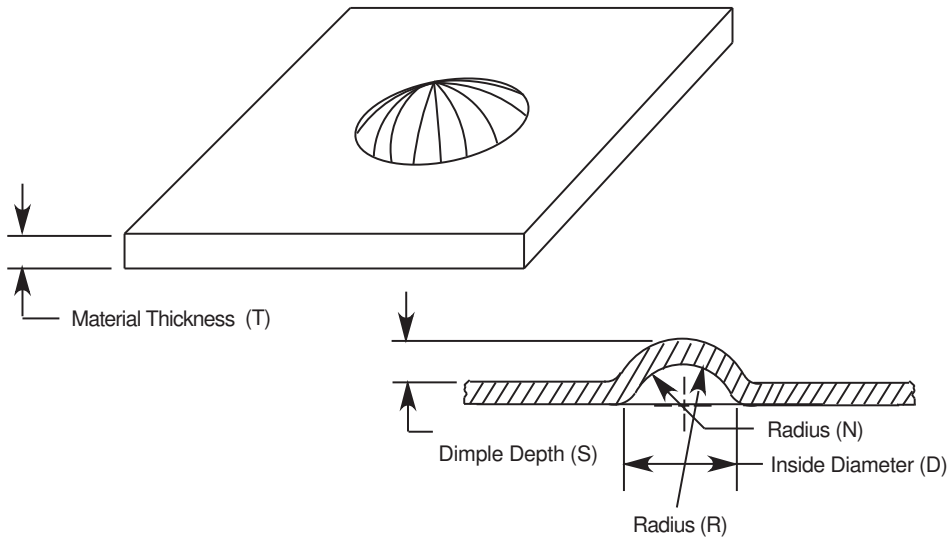
A dimple is a spherical depression that has an inside diameter equal to, or less than, six times the material thickness for low carbon steel (three times the material thickness for aluminum). The depth should be equal to, or less than, one-half the inside diameter of the dimple. See Figure 18, Dimple.

REASON: A spherical depression with an inside diameter greater than indicated by the rule is classified as a rib or embossment (form) feature. See Figure 18, Dimple.

ALTERNATIVE: See subhead 2.2 forms for alternative methods of offsetting sheet metal.

ALTERNATIVE IMPACT: Positive or negative impacts of various form features have to be evaluated based upon specific design requirements. See a Producibility Engineer.

Figure 18: Dimple



Rules:

D = 3T (maximum) for aluminum
D = 6T (maximum) for low carbon steel
S (maximum) = 1/2D

2.2.8 DIMPLE TO HOLE

GUIDELINE

RULE:

The minimum recommended distance that the center line of a dimple should be from the edge of a hole is three times the material thickness plus the radius of the dimple. See Figure 19, Dimple to Hole.

REASONS: Dimples positioned too close to a hole may result in hole distortion or deformation. See Figure 19, Dimple to Hole.

Distortion could lead to manufacturing and/or quality problems.

ALTERNATIVES: Dimples may be located much closer to a hole than the rule suggests by using other manufacturing processes, for example, creating the hole after dimpling.

Dimples may be located much closer to a hole than the rule suggests through the use of relief cuts/slots.

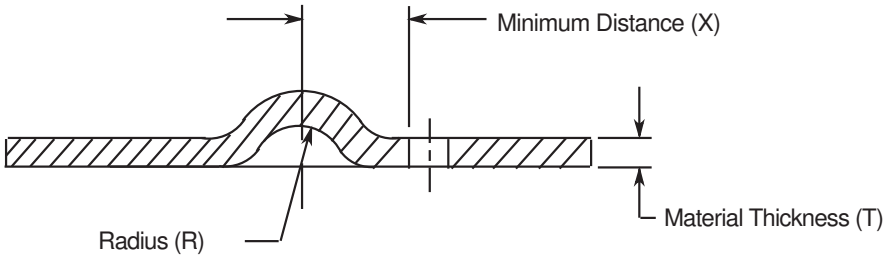
If the design is not impaired by hole distortion, a note could be added to the drawing /database stating that hole distortion is allowable (be specific on location).

ALTERNATIVE IMPACT: These other methods of producing holes require special tooling and/or secondary operations that lead to increased process and/or tooling costs.

Positive or negative impacts of adding a form relief have to be evaluated based upon specific design requirements. See a Producibility Engineer.

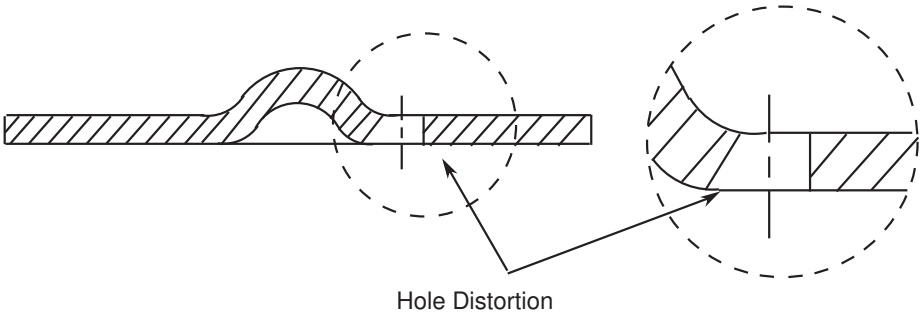
Figure 19: Dimple to Hole

A. Dimple To Hole



Rule: X (minimum) = 3 T + R

B. Hole Distortion



2.2.9 DIMPLE TO EDGE

GUIDELINE

RULE:

The minimum recommended distance that the center line of a dimple should be from the edge is four times the material thickness plus the inside radius of the dimple. See a Producibility Engineer for specific recommendations. See Figure 20, Dimple to Edge.

REASONS: Dimples positioned too close to an edge may result in the distortion /deformation of the edge.

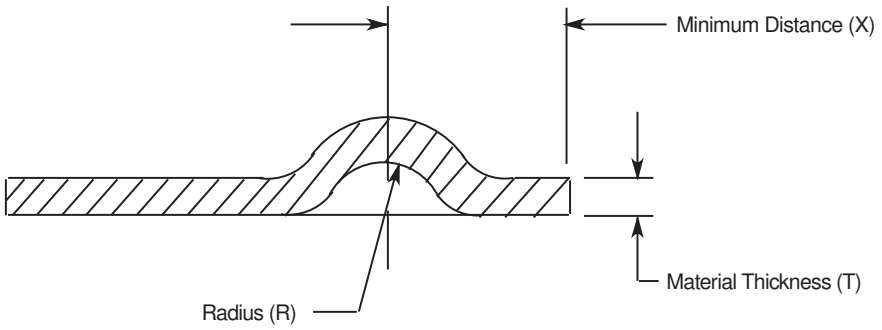
Distortion can lead to manufacturing and/or quality problems.

ALTERNATIVES: Dimples may be located much closer to an edge than the rule suggests through the use of other manufacturing processes, for example, cutting off extra edge material after dimpling.

If the design is not impaired by edge distortion, a note could be added to the drawing /database stating that edge distortion is allowable (be specific on location).

ALTERNATIVE IMPACT: These other manufacturing processes of producing dimples close to an edge without distortion require special tooling and/or secondary operations that lead to increased process and tooling costs.

Figure 20: Dimple to Edge



Rule: X (minimum) = 4T + R

2.2.10 DIMPLE TO BEND

GUIDELINE

RULE:

The minimum recommended distance that a dimple should be from a bend is two times the material thickness plus the inside radius of the dimple plus the radius of the bend. See a Producibility Engineer for specific recommendations. See Figure 21, Dimple to Bend.

REASONS: Dimples positioned too close to a bend may result in the deformation/distortion of the bend and/or dimple. They may also require special tooling.

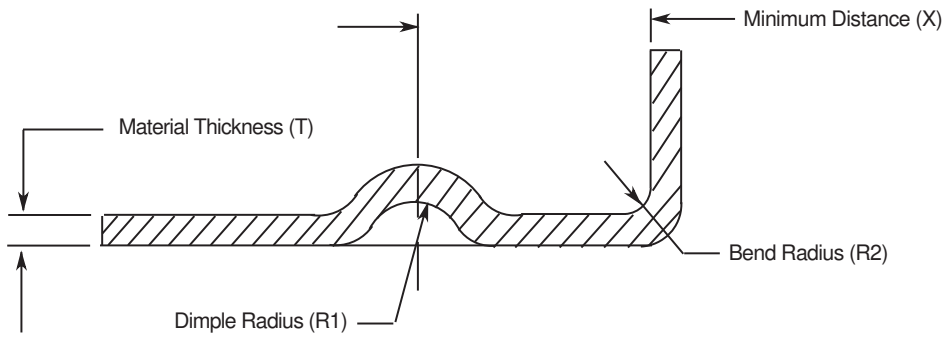
Distortion can lead to manufacturing and/or quality problems.

ALTERNATIVES: Dimples may be located much closer than the rule suggests through the use of relief cuts/slots in the bend.

If the design is not impaired by edge distortion, a note could be added to the drawing /database stating that edge distortion is allowable (be specific on location).

ALTERNATIVE IMPACT: Positive or negative impacts of adding relief cuts/slots have to be evaluated based upon specific design requirements. See a Producibility Engineer.

Figure 21: Dimple to Bend



Rule: X (minimum) = 2T + R1 + R2

2.2.11 DIMPLE TO DIMPLE

GUIDELINE

RULE:

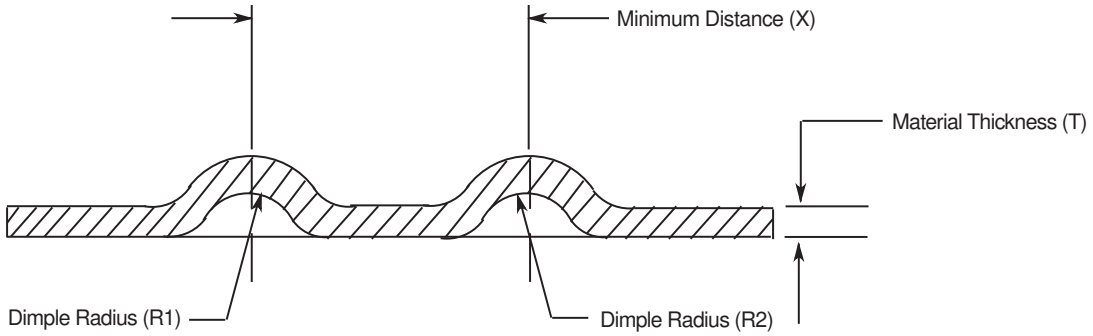
1. The minimum recommended distance that a dimple should be from another dimple is four times the material thickness plus the inside radius of each dimple. See Figure 22, Dimple to Dimple.

REASONS: Dimples positioned too close together on the same plane may result in the distortion of one or both dimples, or warpage (oil-canning) of the material.

ALTERNATIVES: Combine the dimples into a single larger dimple or embossment.

ALTERNATIVE IMPACT: Positive or negative impacts of the alternatives have to be evaluated based upon specific design requirements. See a Producibility Engineer.

Figure 22: Dimple to Dimple



Rule X (minimum) = 4T + R1 + R2

2.2.12 GUSSET

GUIDELINE

RULE:

A gusset is typically used as a method of stiffening or strengthening a bend. The width and depth, recommended at an angle of 45 degrees, is directly proportional to the radius and material thickness. See Figure 23, Gusset.

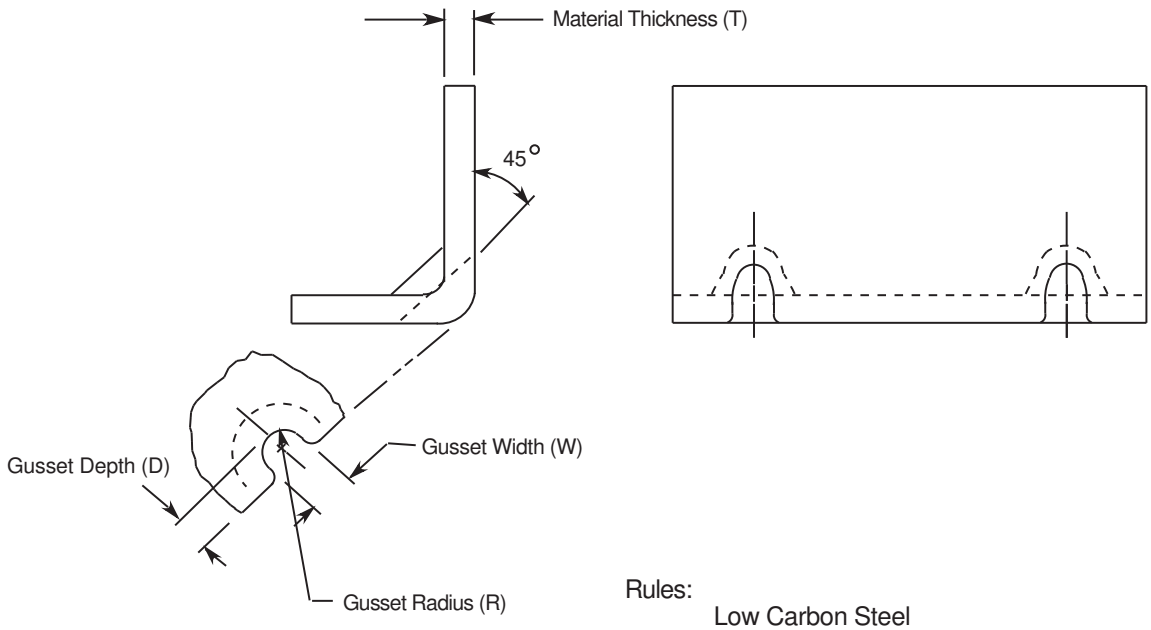
REASON: A gusset produced in excess of the rule does *not* increase the strength of the bend, but can lead to uncontrolled bend flange distortion.

ALTERNATIVES: Change to a welded construction and/or increase the material thickness.

ALTERNATIVE IMPACT: Positive or negative impacts of a welded construction would have to be evaluated based upon specific design requirements. See a Producibility Engineer.

Increased material thickness will lead to increased material cost.

Figure 23: Gusset



Rules:

Low Carbon Steel
- R (minimum) = T
- W (minimum) = 2R
- D (maximum) = 2R
- Recommended Angle = 45°

Aluminum
- R (minimum) = 2T
- W (minimum) = 2R
- D (maximum) = 2R
- Recommended Angle = 45°

2.2.13 GUSSET TO HOLE

GUIDELINE

RULE:

The minimum recommended distance that a gusset should be from the edge of a hole is:

In a parallel plane—eight times the material thickness plus the radius of the gusset.
In a perpendicular plane—four times the material thickness plus the radius of the gusset.

See Figure 24, Gusset to Hole.

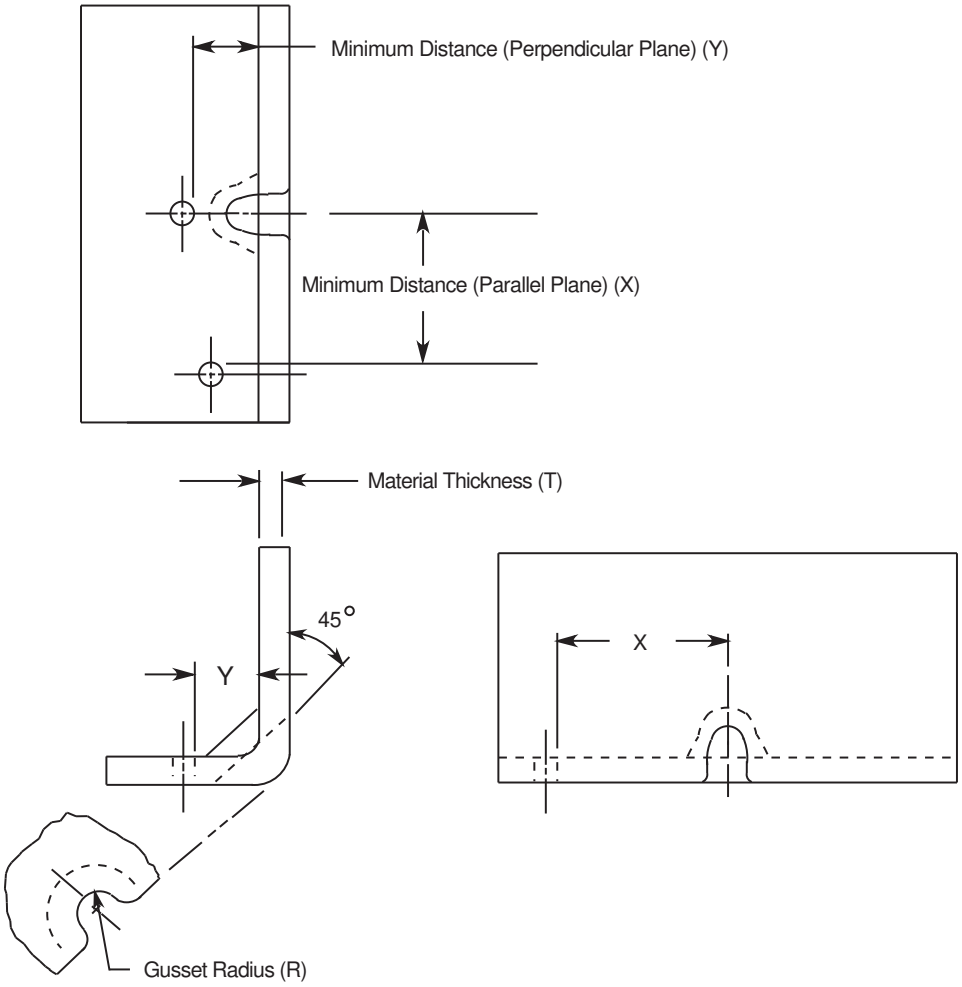
REASON: Holes positioned too close to a gusset may result in hole distortion leading to manufacturing or quality problems.

ALTERNATIVES: Holes may be located much closer to a gusset than the rule suggests by using other manufacturing methods or by using secondary operations, such as to punch after forming.

If the design is not impaired by hole distortion, a note could be added to the drawing /database stating that hole distortion is allowable (be specific on location).

ALTERNATIVE IMPACT: Methods of producing holes other than by punching or secondary operations can lead to increased process and/or tooling costs.

Figure 24: Gusset to Hole



Rules:

X (minimum) = 8T + R
Y (minimum) = 4T + R

2.2.14 GUSSET TO GUSSET

GUIDELINE

RULE:

The minimum recommended distance between two gussets is equal to ten times the material thickness plus the radius of each gusset. See Figure 25, Gusset to Gusset.

REASONS: Gussets positioned too close to one another may result in the distortion or deformation of one or both gussets.

Gussets positioned too close together on the same plane may result in the distortion or warpage (oil-canning) of the material.

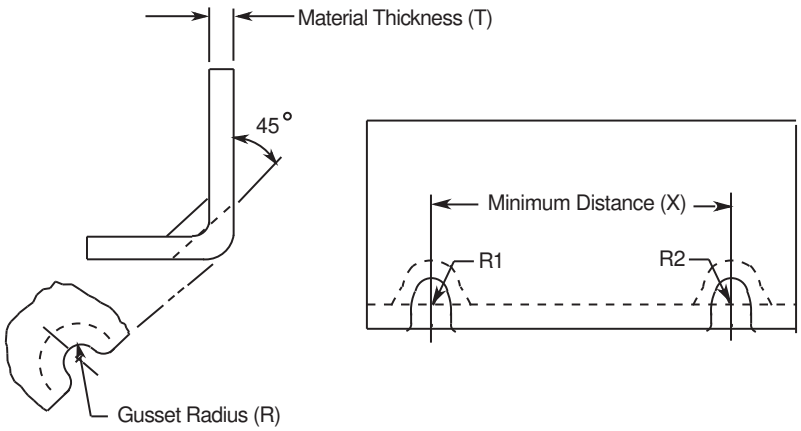
ALTERNATIVES: Change to a welded construction and/or increase the material thickness.

Redesign through the use of additional supporting and strengthening parts.

ALTERNATIVE IMPACT: Positive or negative impacts of a welded construction would have to be evaluated based upon specific design requirements. See a Producibility Engineer.

Increased material thickness and/or additional parts lead to increased process and material costs.

Figure 25: Gusset to Gusset



Rule:
X (minimum) = 10T + R1 + R2

2.2.15 GUSSET TO EDGE

GUIDELINE

RULE:

The minimum recommended distance that a gusset should be from an edge is:

In a parallel plane—ten times the material thickness plus the inside radius of the gusset.
In a perpendicular plane—four times the material thickness plus the inside radius of the gusset.

See Figure 26, Gusset to Edge.

REASON: Gussets positioned too close to an edge may result in the distortion/deformation of the edge leading to manufacturing and/or quality problems.

ALTERNATIVES: Change to a welded construction and/or increase the material thickness.

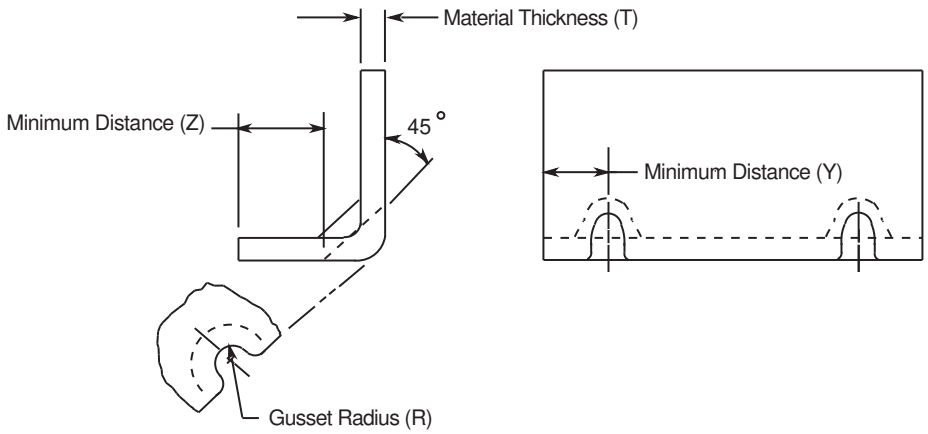
Redesign through the use of additional supporting and strengthening parts.

If the design is not impaired by edge distortion a note can be added to the drawing/database stating that edge distortion is allowable (be specific on location).

ALTERNATIVE IMPACT: Positive or negative impacts of a welded construction would have to be evaluated based upon specific design requirements. See a Producibility Engineer.

Increased material thickness and/or additional parts leads to increased process and material costs.

Figure 26: Gusset to Edge



Rules:

Y (minimum) = 10T + R
Z (minimum) = 4T + R

2.2.16 GUSSET TO BEND

INFORMATION

RULE: N/A... A Gusset is part of a Bend (Form) feature.

2.2.17 CURL

GUIDELINE

RULE:

A curl is a rolled over edge condition typically used as a means of joining two or more parts in the form of a hinge. The curl has a minimum inside radius equal to two times the material thickness, to a minimum opening (gap) equal to one material thickness. See Figure 27, Curl.

NOTE

Curls are not recommended for aluminum or stainless steel due to fracturing. See a Producibility Engineer.

REASONS: If a curl is formed with a radius less than the rule suggests, fracturing of the material may occur. See Figure 27, Curl.

Forming a curl with an opening (gap) of less that one material thickness has no structural benefit and may be more difficult to manufacture.

ALTERNATIVES: There are other methods of producing a curl feature on a part, such as in the use of multiple parts (purchased hinge) and assembly techniques (such as rivets or welding). See Figure 27, Curl.

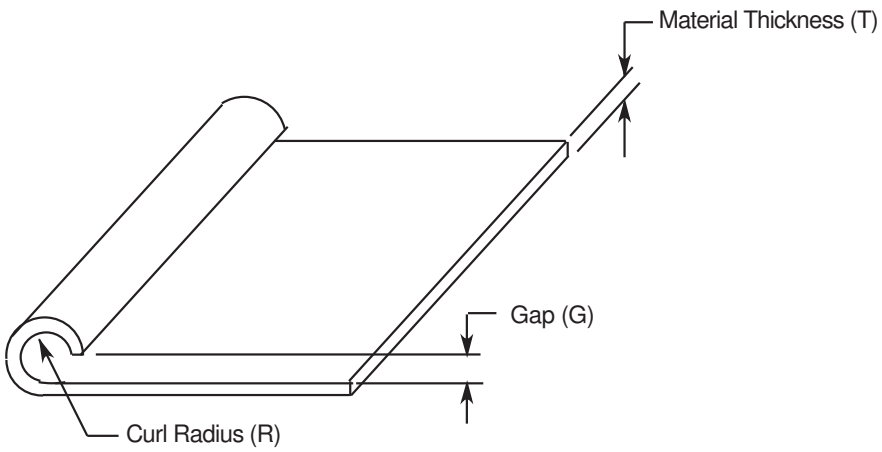
Curls may be formed without leaving an opening (no gap, closed). See Figure 27, Curl.

ALTERNATIVE IMPACT: Positive or negative impacts of an assembly (for example, welded construction) have to be evaluated based upon specific design requirements. See a Producibility Engineer.

The closing of a curl may require special tooling and/or secondary operations that lead to increased process and/or tooling costs.

Figure 27: Curl

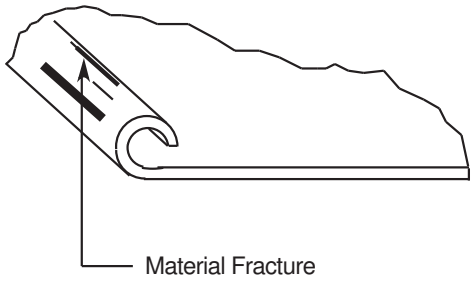
Curl



Rules:

R (minimum) = 2T
G (minimum) = T

Curl Fracture



2.2.18 CURL TO HOLE

GUIDELINE

RULE:

The minimum recommended distance from a curl to the edge of a hole is one times the material thickness plus the radius of the curl. See Figure 28, Curl to Hole.

NOTE

Curls are not recommended for aluminum or stainless steel due to fracturing. See a Producibility Engineer.

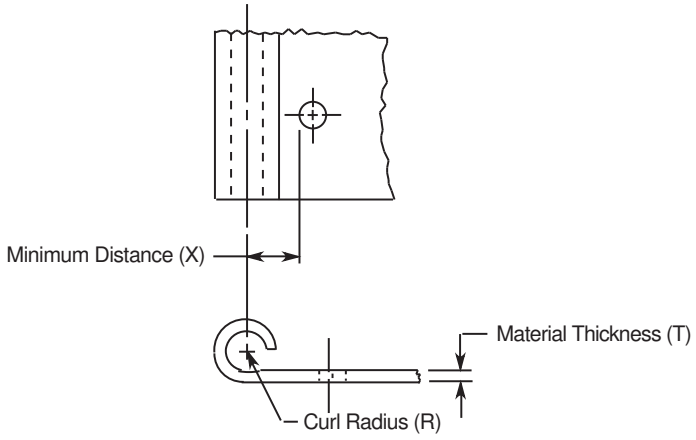
REASON: Holes positioned too close to a curl may result in hole distortion leading to manufacturing and/or quality problems.

ALTERNATIVES: Holes may be located much closer to the curl than the rule suggests by using other manufacturing methods (for example, drilling or by using secondary operations, such as to punch after forming).

If the design is not impaired by hole distortion, a note can be added to the drawing/database stating that hole distortion is allowable (be specific on location).

ALTERNATIVE IMPACT: These other methods of producing holes require special tooling and/or secondary operations that lead to increased process and/or tooling costs.

Figure 28: Curl to Hole



Rule: X (minimum) = T + R

2.2.19 CURL TO BEND

GUIDELINE

RULE:

The minimum recommended distance from a bend to a curl is:

Internal bend—six times the material thickness plus the inside radius of the curl. External bend–nine times the material thickness plus the inside radius of the curl.

See Figure 29, Curl to Bend.

NOTE

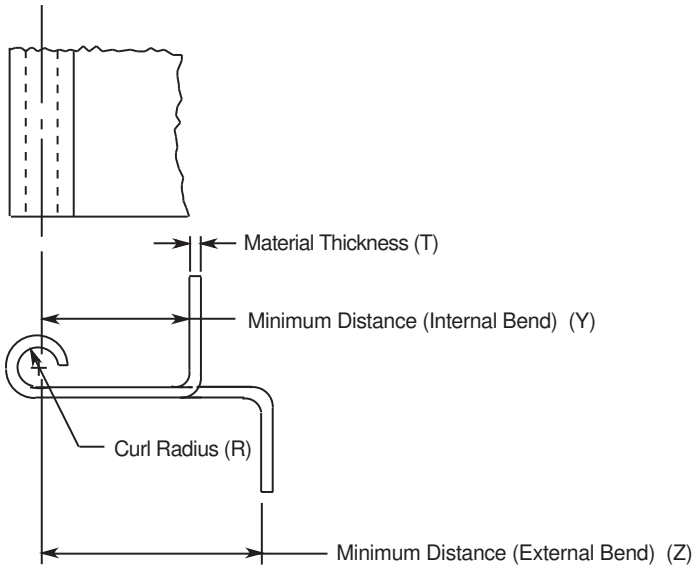
Curls are not recommended for aluminum or stainless steel due to fracturing. See a Producibility Engineer.

REASON: Curls positioned too close to a bend may be *very* difficult to produce when using standard manufacturing processes and/or tooling.

ALTERNATIVE: Curls may be located closer than the rule suggests by using special processes and/or tooling.

ALTERNATIVE IMPACT: These special manufacturing processes and/or tooling lead to greatly increased process and/or tooling costs, along with manufacturability and/or quality problems.

Figure 29: Curl to Bend



Y (minimum) = 6T + R
Z (minimum) = 9T + R

2.2.20 CURL TO CURL

GUIDELINE

RULE:

The minimum recommended distance between two curls has to be evaluated based upon specific design requirements. See a Producibility Engineer for specific recommendations.

NOTE

Curls are not recommended for aluminum or stainless steel due to fracturing. See a Producibility Engineer.

REASON: Curls positioned too close to each other may result in the deformation and distortion of the curl(s). This may also require special tooling.

ALTERNATIVES: There are other methods of producing curls features close to each other, such as using multiple parts (curl in each) and assembly techniques (such as rivets or welding).

ALTERNATIVE IMPACT: Positive or negative impacts of an assembly (for example, welded construction) have to be evaluated based upon specific design requirements. See a Producibility Engineer.

2.2.21 HEM

INFORMATION

RULES:

1. A hem is typically used to eliminate sharp edges (such as a grommet to eliminate a burr condition) or to provide additional stiffness (such as to support and strengthen) to the edge of a surface. See a Producibility Engineer for specific recommendations. See Figure 30, Hem.
2. Hemmed edges are classified according to three variations/shapes. See Figure 30, Hem.
   - Teardrop method (preferred): easiest to manufacture with fewest finishing problems.
   - Open (alternative to teardrop): more difficult to manufacture.
   - Closed (least-preferred method): creates many finishing problems.

REASONS: Hems may be economically produced by standard manufacturing processes and/or tooling.

Hemmed edges are produced in various shapes to provide flexibility in meeting design and/or manufacturability requirements.

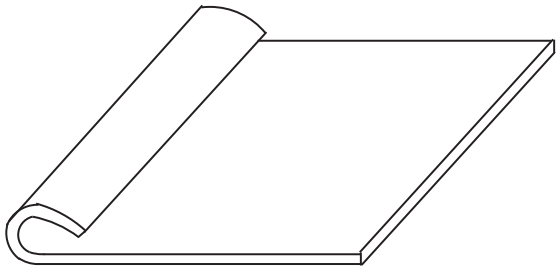
ALTERNATIVES: Hems may be simulated through the use of additional (fabricated and/or purchased) parts. Examples include, but are not limited to:

- Purchased grommets (for edge burr condition)
- Small offset (for edge stiffening)
- Weldment (additional parts for edge stiffening)
- Angle bends (for edge burr condition and/or stiffening)

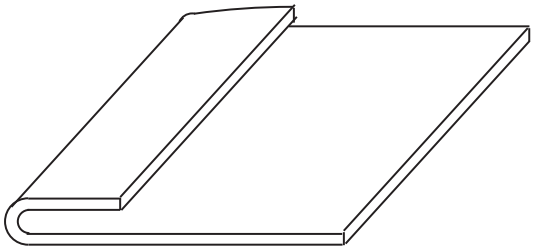
ALTERNATIVE IMPACT: Positive or negative impacts of a fabricated or purchased hem feature must be evaluated based upon specific design requirements. See a Producibility Engineer.

Figure 30: Hem

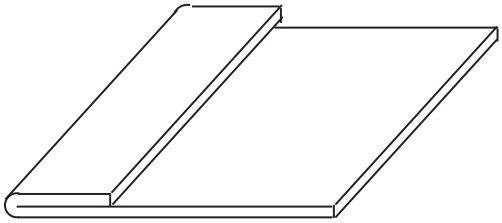
A. Teardrop Hem



B. Open Hem



C. Closed Hem



2.2.22 HEM - TEARDROP

GUIDELINE

RULES:

1. A teardrop hem is a formed feature with a minimum diameter equal to the material thickness. Return flange height (measured from outside the bend and always bent greater than 180 degrees) is equal to or greater than four times the material thickness, up to 305.0 mm (12 inches) in bend length. A flange height of 12.5 mm (0.50 inch) for bend lengths of 305.0 mm (12 inches) or greater is recommended.
2. The recommended minimum hem opening should be equal to one quarter of the material thickness. See Figure 31, Hem - Teardrop.

NOTE

For this rule to apply, the material thickness should be less than 3.20 mm (0.125 inch).

Hems are not recommended for aluminum or stainless steel due to fracturing. See a Producibility Engineer.

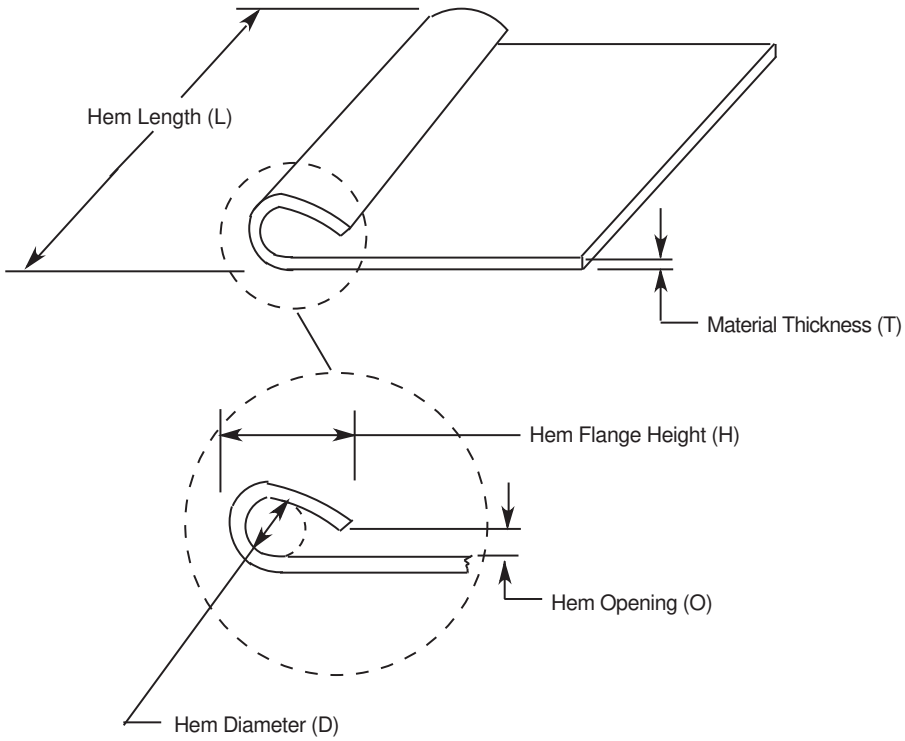
REASON: Forming a hem with a radius and/or flange height less than the rule suggests, may lead to manufacturability (tooling and process) and/or quality problems (flange radius fracturing or squareness).

ALTERNATIVES: Hems may be simulated by using additional fabricated and/or purchased parts. Examples include, but are not limited to:

- Purchased grommets (for edge burr condition)
- Small offset (for edge stiffening)
- Weldment (additional parts for edge stiffening)
- Angle bends (for edge burr condition and/or stiffening)

ALTERNATIVE IMPACT: Positive or negative impacts of a fabricated or purchased hem features have to be evaluated based upon specific design requirements. See a Producibility Engineer.

Figure 31: Hem - Teardrop



Rules:

O (minimum) = 1/4 T
D (minimum) = T
T (maximum) = 3.2 mm (.125 inch)

When L is 305.0 mm (12 inch) or less, then H (minimum) = 4T
When L is greater than 305.0 mm (12 inch), then H (minimum) = 12.5 mm (1/2 inch)

2.2.23 HEM - OPEN

GUIDELINE

RULE:

An open hem is a formed feature with a minimum inside diameter equal to the material thickness. Return flange height is measured from the outside of the bend and is always bent at 180 degrees.

The minimum return flange height for bend lengths less than 305.0 mm (12 inches) is equal to or greater than 4 times the material thickness. The minimum return flange height for bend lengths equal to or greater than 305.0 mm (12 inches) is 12.5 mm (0.50 inch). See Figure 32, Hem - Open.

NOTE

For this rule to apply, the material thickness should be less than 3.20 mm (0.125 inch).

Hems are not recommended for aluminum or stainless steel due to fracturing. See a Producibility Engineer.

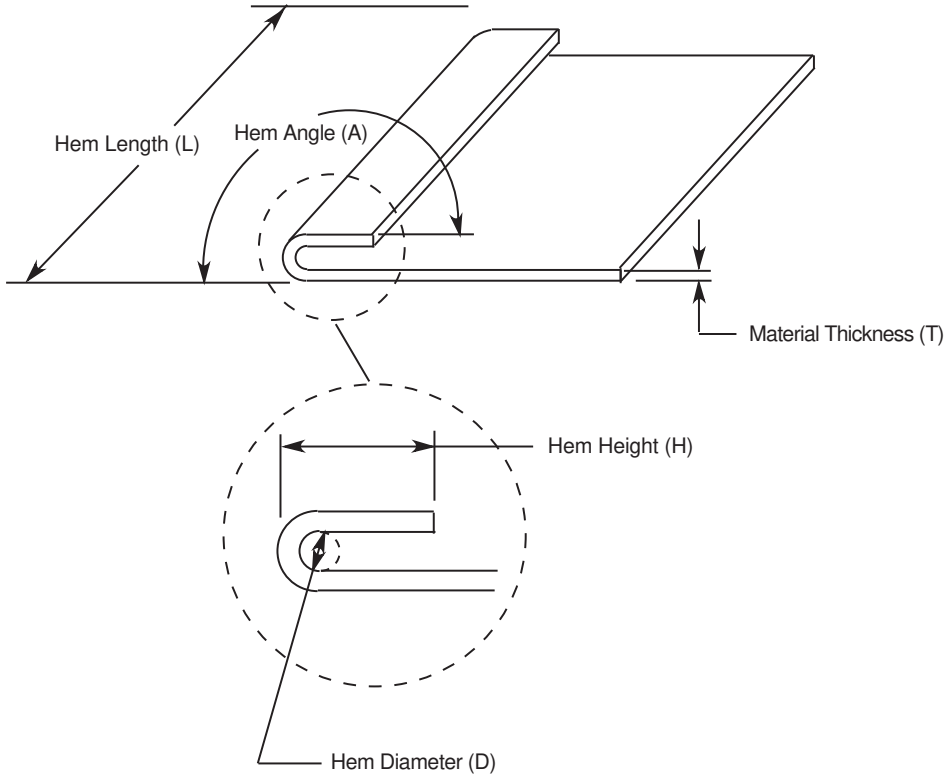
REASON: Forming a hem with a radius and/or flange height less than the rule suggests may lead to manufacturability (tooling and process) and/or quality problems (flange radius fracturing or squareness).

ALTERNATIVES: Hems may be simulated by using additional fabricated or purchased parts. Examples include, but are not limited to:

- Purchased grommets (for edge burr condition)
- Small offset (for edge stiffening)
- Weldment (additional parts for edge stiffening)
- Angle bends (for edge burr condition or stiffening)

ALTERNATIVE IMPACT: Positive or negative impacts of a fabricated or purchased hem features have to be evaluated based upon specific design requirements. See a Producibility Engineer.

Figure 32: Hem - Open



Rules:

A = 180°
T (maximum) = 3.2 mm (0.125 inch)
D (minimum) = T

When L is 305.0 mm (12 inches) or less, then H (minimum) = 4T .
When L is greater than 305.0 mm (12 inches), then H (minimum) = 12.5 mm (1/2 inch) .

2.2.24 HEM - CLOSED

GUIDELINE

RULE:

A closed hem is a formed feature with an inside radius equal to zero. Return flange height (measured from outside the bend) is equal to or greater than four times the material thickness, up to 305.0 mm (12 inches) in bend length or a minimum flange height of 12.5 mm (0.5 inch) for bend lengths of 305.0 mm (12 inches) or greater. See Figure 33, Hem - Closed.

NOTE

For this rule to apply, the material thickness should be less than 3.20 mm (0.125 inch).

A closed hem will always lead to a degree of fracturing of the material at the bend. It also leads to entrapment of solutions during the finishing process. See Figure 33, Hem - Closed.

Hems are not recommended for aluminum or stainless steel due to fracturing. See a Producibility Engineer.

REASON: Forming a hem with an inside radius of zero and flange height less than the rule suggests may lead to manufacturability (tooling and process) and/or quality problems (flange radius fracturing or squareness).

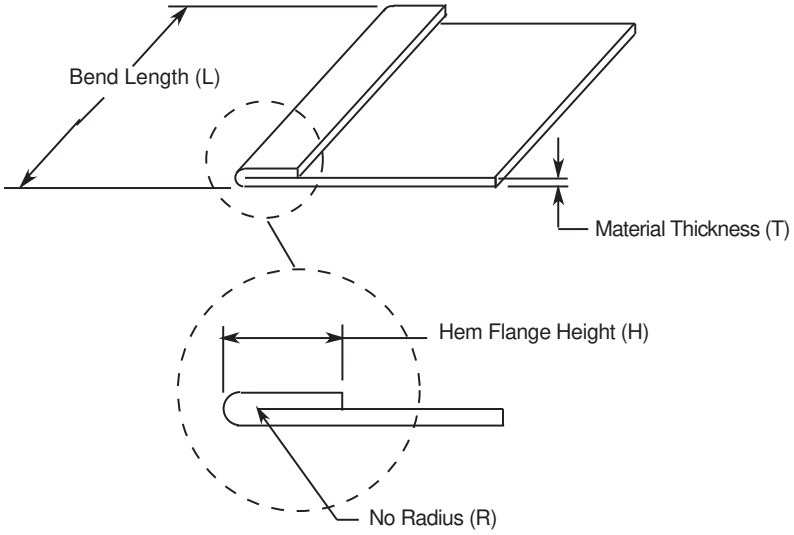
ALTERNATIVES: Hems may be simulated by using additional fabricated and/or purchased parts. Examples include, but are not limited to:

- Purchased grommets (for edge burr condition)
- Small offset (for edge stiffening)
- Weldment (additional parts for edge stiffening)
- Angle bends (for edge burr condition and/or stiffening)

ALTERNATIVE IMPACT: Positive or negative impacts of a fabricated or purchased hem feature would have to be evaluated based upon specific design requirements. See a Producibility Engineer.

Figure 33: Hem - Closed

A. Closed Hem

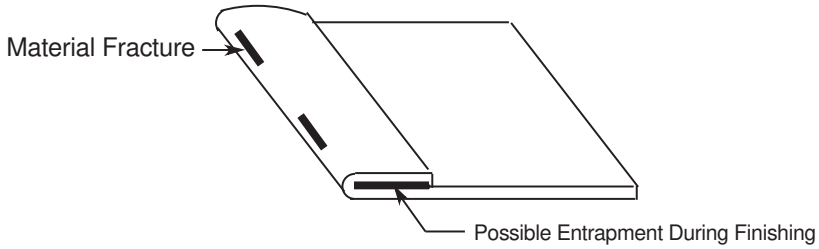


Rules:

T (maximum) = 3.2 mm (0.125 inch)

When L is 12 inches (305.0 mm) or less, then H (minimum) = 4T .
When L is greater than 305.0 mm (12 inches), then H (minimum) = 12.5 mm (1/2 inch) .

B. Hem Fracture



2.2.25 HEM TO HOLE

GUIDELINE

RULE:

The minimum recommended distance from the edge of a hole to the outside edge of a hem is two times the material thickness plus the radius of the hem. See Figure 34, Hem to Hole.

NOTE

Hems are not recommended for aluminum or stainless steel due to fracturing. See a Producibility Engineer.

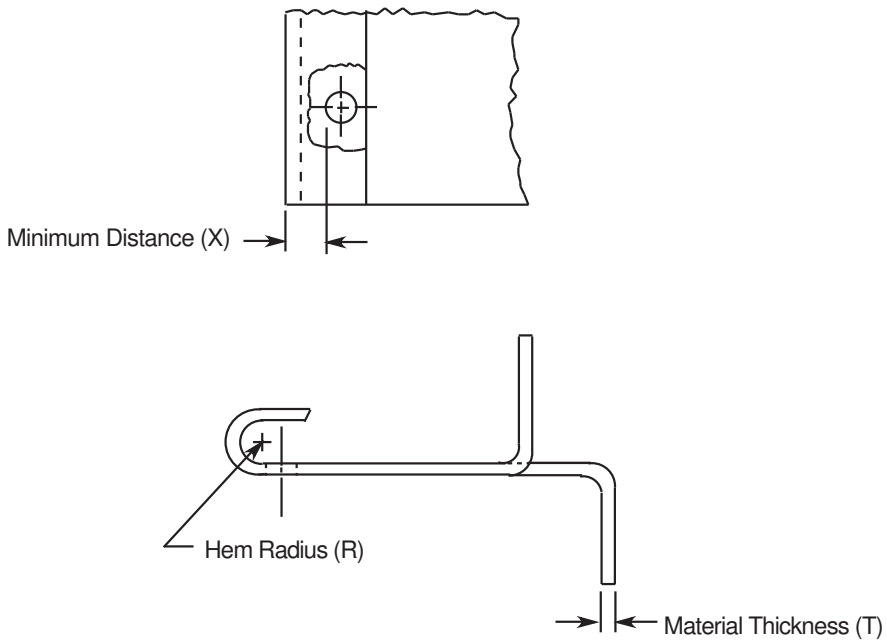
REASON: Holes positioned too close to the hem radius may result in hole distortion that leads to manufacturing and/or quality problems.

ALTERNATIVES: Holes may be located much closer to the hem radius than the rule suggests by using other manufacturing methods. See a Producibility Engineer for specific recommendations.

If the design is not impaired by hole distortion, a note can be added to the drawing/database stating that hole distortion is allowable (be specific on location).

ALTERNATIVE IMPACT: These other methods of producing holes require special tooling and/or secondary operations that lead to increased process and/or tooling costs.

Figure 34: Hem to Hole



X (minimum) = 2T + R

2.2.26 HEM TO EDGE

INFORMATION

RULE N/A... A hem is a condition of an edge feature.

2.2.27 HEM TO BEND

GUIDELINE

RULE:

The minimum recommended distance from a hem to a bend is:

Internal bend — five times the material thickness.
External Bend — eight times the material thickness.
See Figure 35, Hem to Bend.

NOTE

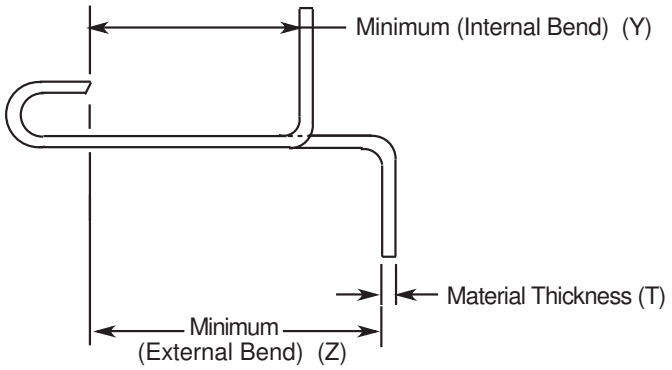
Hems are not recommended for aluminum or stainless steel due to fracturing. See a Producibility Engineer.

REASON: Hems positioned too close to a bend may be *very* difficult to produce when using standard manufacturing processes and/or tooling.

ALTERNATIVE: Hems may be located closer than the rule suggests by using special processes and/or tooling.

ALTERNATIVE IMPACT: These special manufacturing processes and/or tooling lead to greatly increased process and/or tooling costs, along with manufacturability and/or quality problems.

Figure 35: Hem to Bend



Rules:

Y (minimum) = 5T

Z (minimum) = 8T

2.2.28 HEM TO HEM

GUIDELINE

RULE:

The minimum recommended distance between two hems has to be evaluated based upon specific design requirements. See a Producibility Engineer for specific recommendations.

NOTE

Hems are not recommended for aluminum or stainless steel due to fracturing. See a Producibility Engineer.

REASON: Hems positioned too close to each other may result in the deformation/distortion of the hem. They may also require special tooling.

ALTERNATIVES: There are other methods of producing hem features close together, such as in the use of multiple parts (hems in each) and assembly techniques that require rivets or welding.

ALTERNATIVE IMPACT: Positive or negative impacts of an assembly (for example, welded construction) have to be evaluated based upon specific design requirements. See a Producibility Engineer.

2.2.29 RIB

GUIDELINE

RULE:

A rib is a cylindrical depression, similar to a dimple, and used to strengthen or stiffen a surface. It has a maximum inside radius equal to three times the material thickness to a maximum depth equal to the inside radius. There are no constraints on the length. See Figure 36, Rib.

REASON: A depression, spherical or otherwise, with an inside radius greater than the rule specifies is classified as an embossment (form) feature. See Figure 36, Rib.

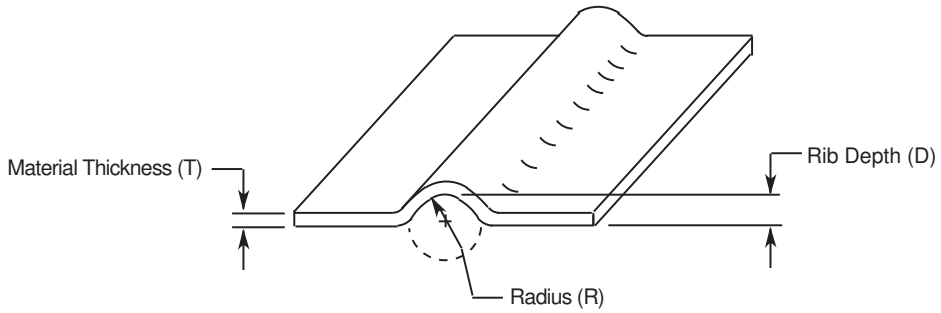
ALTERNATIVES: Refer to subhead 2.2 *Forms* for alternative methods of stiffening or strengthening sheet metal.

There are other methods of stiffening or strengthening sheet metal, such as using other (or thicker) materials or multiple parts (assembly).

ALTERNATIVE IMPACT: Positive or negative impacts of various form features have to be evaluated based upon specific design requirements. See a Producibility Engineer.

Positive or negative impacts of different, that is thicker, materials and/or multiple parts (assembly) have to be evaluated based upon specific design requirements. See a Producibility Engineer.

Figure 36: Rib



Rule:

R (maximum) = 3T
D (maximum) = R

2.2.30 RIB TO HOLE

GUIDELINE

RULE:

The minimum recommended distance of the center line of a rib to the edge of a hole is three times the material thickness plus the radius of the rib. See Figure 37, Rib to Hole.

REASON: Ribs positioned too close to a hole may result in hole distortion or deformation leading to manufacturing and/or quality problems. See Figure 37, Rib to Hole.

ALTERNATIVES: Ribs may be located much closer to a hole than the rule suggests through the use of relief cuts/slots. They may also be located closer to a hole by using other manufacturing processes, such as machine hole after forming.

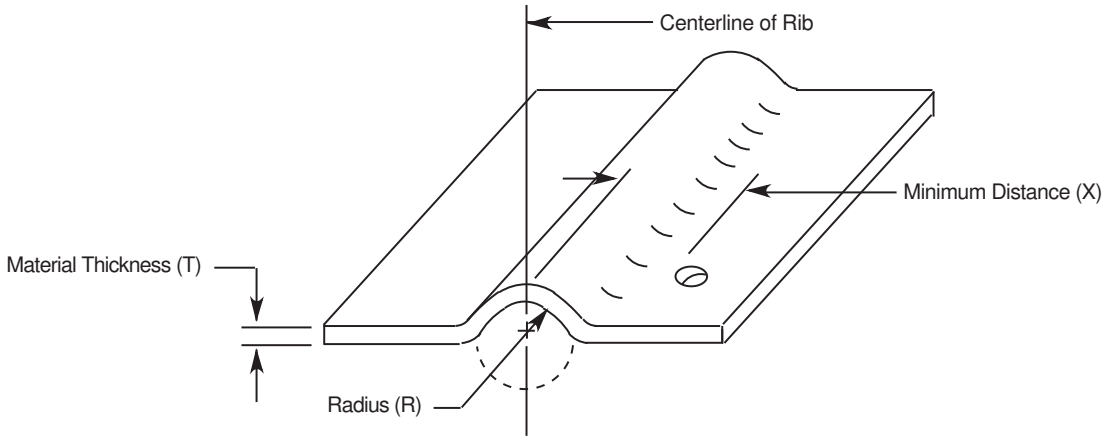
If the design is not impaired by hole distortion, a note can be added to the drawing/database stating that hole distortion is allowable (be specific on location).

ALTERNATIVE IMPACT: These other methods of producing holes require special tooling and/or secondary operations leading to increased process and/or tooling costs.

Positive or negative impacts of adding form reliefs have to be evaluated based upon specific design requirements. See a Producibility Engineer.

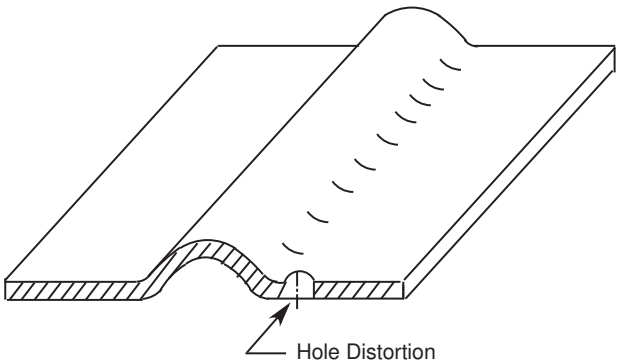
Figure 37: Rib to Hole

A. Rib To Hole



Rule:
X (minimum) = 3T + R

B. Hole Distortion



2.2.31 RIB TO EDGE

GUIDELINE

RULE:

The minimum recommended distance from a rib to the edge is:

In a perpendicular plane — four times the material thickness plus the inside radius of the rib.

NOTE

Once you go beyond the rule limitations, the rib must extend to the edge.

In a parallel plane — eight times the material thickness plus the inside radius of the rib. See Figure 38, Rib to Edge.

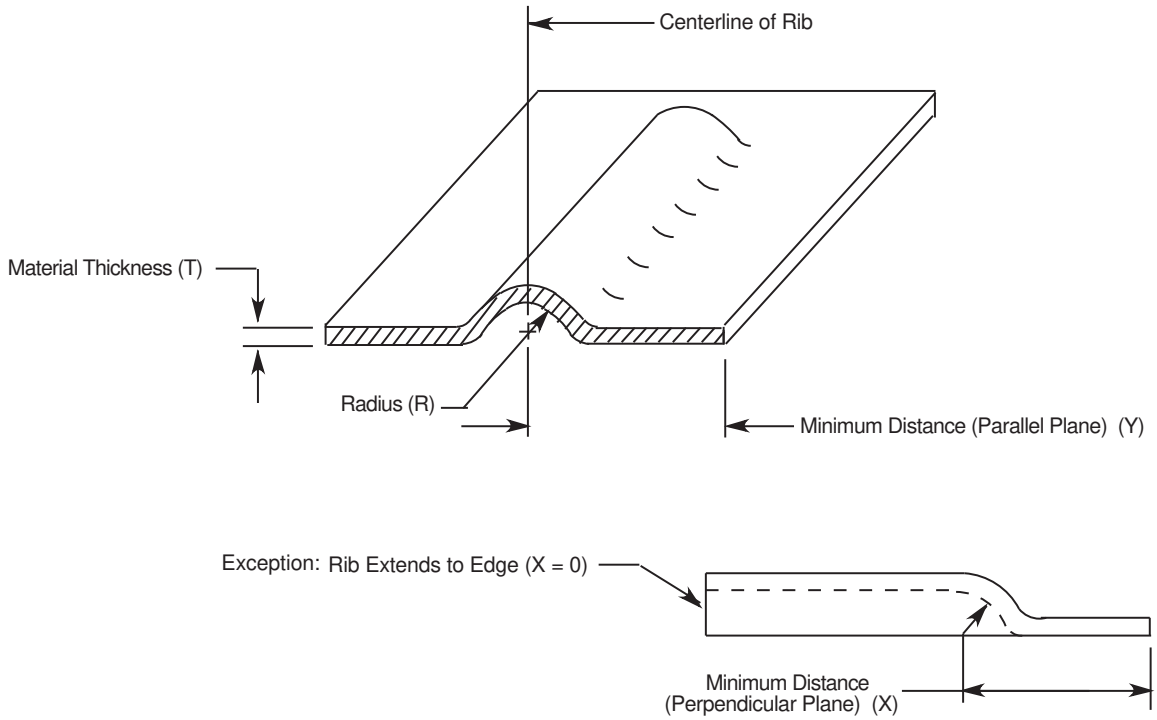
REASON: Ribs positioned too close to an edge may result in the distortion or deformation of the edge leading to manufacturing and/or quality problems.

ALTERNATIVES: Ribs may be located much closer to an edge than the rule suggests through the use of other manufacturing processes (for example, cutting off extra edge material after forming ribs).

If the design is not impaired by edge distortion, a note can be added to the drawing/database stating that edge distortion is allowable (be specific on location).

ALTERNATIVE IMPACT: These other manufacturing processes of producing ribs close to an edge without distortion require special tooling and/or secondary operations that lead to increased process and tooling costs.

Figure 38: Rib to Edge



Rules:

X (minimum) = 4T + R (or 0, if extends to edge)
Y (minimum) = 8T + R

2.2.32 RIB TO BEND

GUIDELINE

RULES:

The minimum recommended distance from a rib to a bend is:

With bend perpendicular to the rib —

Low Carbon Steel — two times the material thickness plus the inside radius of the rib, plus the radius of the bend.
Aluminum or stainless steel — four times the material thickness plus the inside radius of the rib, plus the radius of the bend.
See a Producibility Engineer for specific recommendations. See Figure 39, Rib to Bend.

With bend parallel to the rib — see a Producibility Engineer.

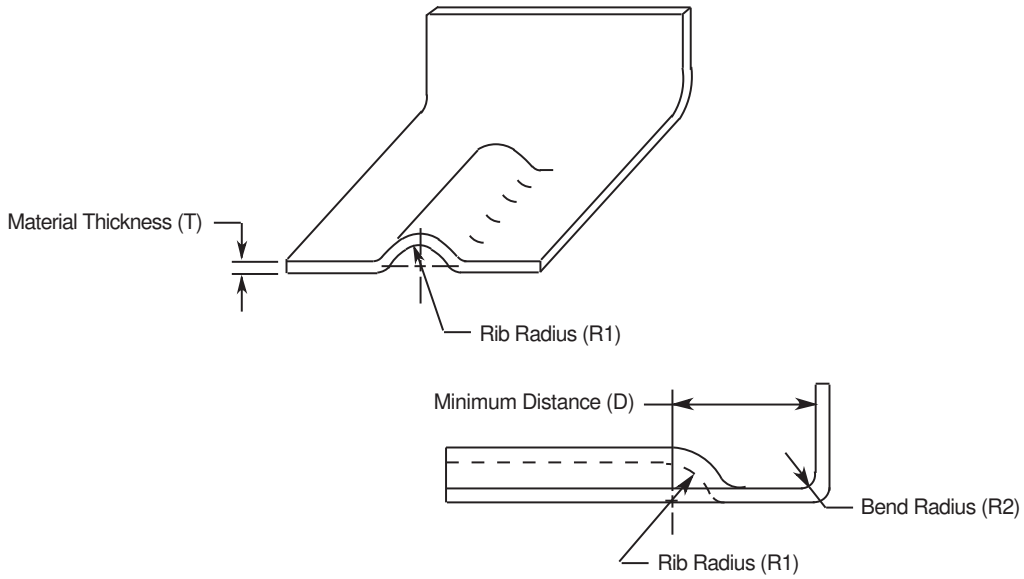
REASON: Ribs positioned too close to a bend may result in the deformation and distortion of the bend and/or rib. They may also require the need of special tooling. See Figure 39, Rib to Bend.

ALTERNATIVE: Ribs may be located much closer to a bend than the rule suggests through the use of relief cuts/slots. See Figure 39, Rib to Bend.

ALTERNATIVE IMPACT: Positive or negative impacts of adding relief cuts/slots have to be evaluated based upon specific design requirements. See a Producibility Engineer.

Figure 39: Rib to Bend

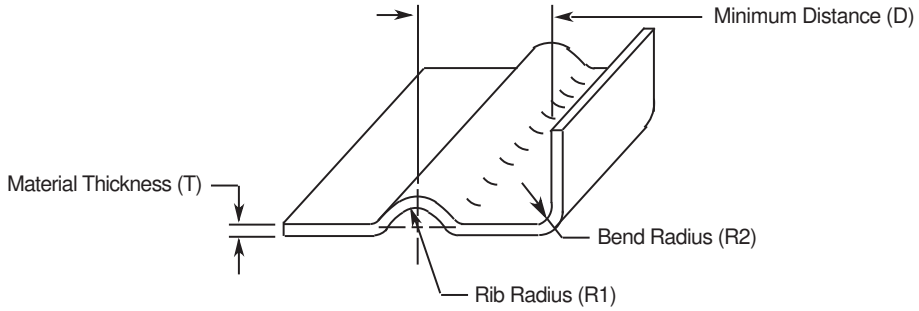
A. Rib Perpendicular To Bend



Rule:

D (minimum) = 2T + R1 + R2 for low carbon steel
= 4T + R1 + R2 for aluminum
= 4T + R1 + R2 for stainless steel

B. Rib Parallel To Bend



Rule:

For minimum distance D, see Producibility Engineering for specific design recommendation.

2.2.33 RIB TO RIB

GUIDELINE

RULE:

The minimum recommended distance between two parallel ribs is ten times the material thickness plus the radius of each rib.

See Figure 40, Rib to Rib.

REASONS: Ribs positioned too close to each another may result in the distortion or deformation of one or both ribs.

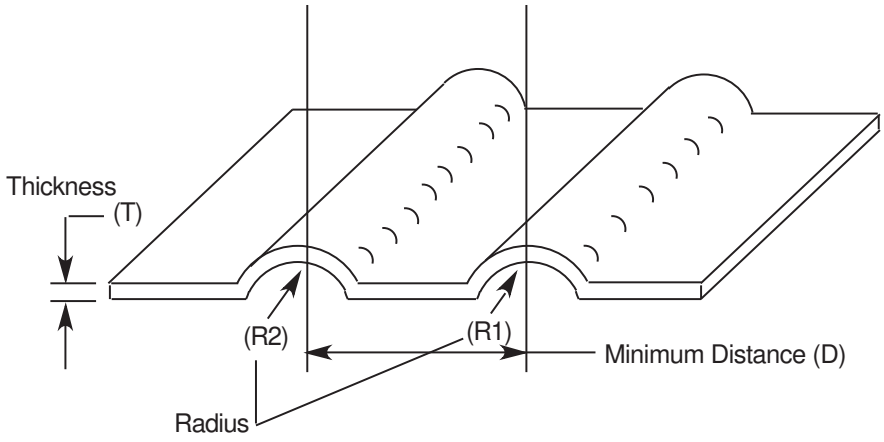
Ribs positioned too close together (on the same plane) may result in the distortion or warpage (oil-canning) of the material.

ALTERNATIVES: Combine the ribs into a single larger embossment.

Change to a fabricated (or welded) construction and/or increase the material thickness.

ALTERNATIVE IMPACT: Positive or negative impacts of alternatives have to be evaluated based upon specific design requirements. See a Producibility Engineer.

Figure 40: Rib to Rib



Rule: D (minimum) = 10T + R1 + R2

2.2.34 EMBOSSMENT

INFORMATION

RULES:

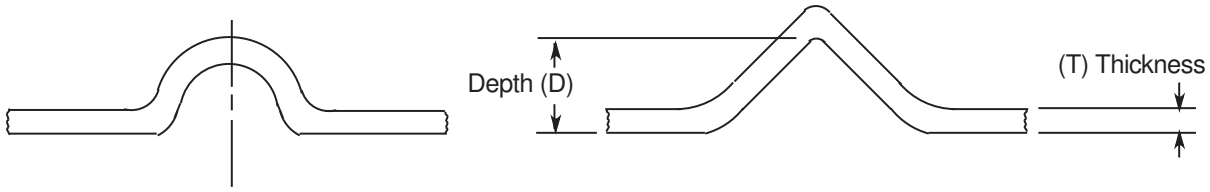
1. An embossment is a depression or a drawing to a shallow depth.
2. An embossment is defined as a depression greater than a rib or dimple, but less than a draw. See Figure 41, Embossment.
3. An embossment may be produced in various shapes. See Figure 41, Embossment, for preferred dimensioning. Examples include, but are not limited to:
   - Round embossments
   - V embossments
   - Interior offset embossments

REASON: Embossments are difficult to define because they are not limited to a particular shape or size. Specific design requirements should be reviewed with a Producibility Engineer for manufacturing considerations.

ALTERNATIVES: See rib, dimple or drawn features.

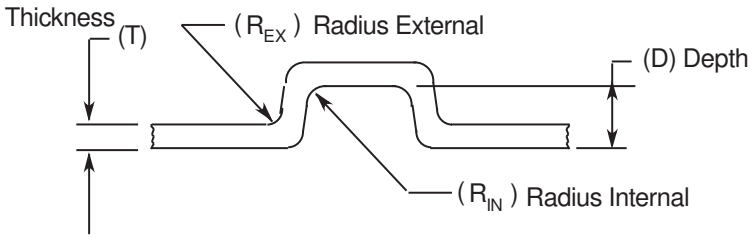
ALTERNATIVE IMPACT: Positive or negative impacts of any variation of a form (for example, rib, dimple, or draw), have to be evaluated based upon specific design and manufacturing requirements or constraints. See a Producibility Engineer.

Figure 41: Embossment



Round Embossment

V Embossment
D (maximum) = 3T



Interior Offset
D (maximum) = R_{EX} + R_{IN}

2.2.35 EMBOSSMENT SHAPES

GUIDELINE

RULES:

1. An embossment may be produced in various shapes and sizes. The maximum depth is proportional to the internal radius or material thickness. See Figure 42, Embossment Shapes.
2. Flat embossment — maximum depth is equal to the internal radius plus the external radius.
3. V embossment — maximum depth is equal to three times the material thickness.

NOTE

Due to the potential variation in size and shape of embossments relative to a particular design requirement, *rules* for interrelationships between features may *not* be created. See a Producibility Engineer for recommendations.

REASONS: Embossments improperly configured may lead to manufacturability problems such as:

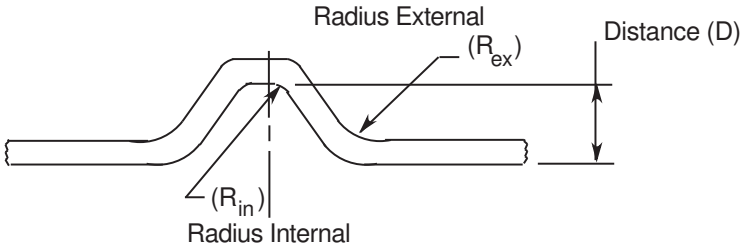
- Material distortion or deformation (oil-canning)
- Tearing or cracking of material
- Distortion or deformation of other features

ALTERNATIVES: There are other methods of forming the surface that result in less material distortion, such as the use of dimples, ribs or multiple part assembly.

ALTERNATIVE IMPACT: Positive or negative impacts of various form features or multiple parts have to be evaluated based upon specific design requirements. See a Producibility Engineer.

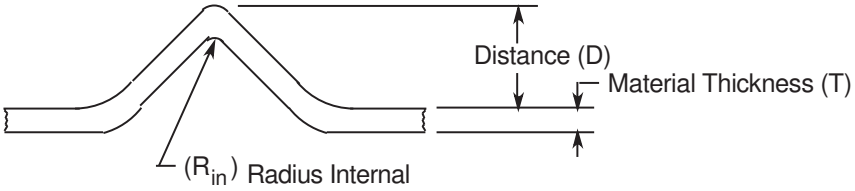
Figure 42: Embossment Shapes

Flat Embossment



Rule:
D (maximum) = R_{in}+ R_{ex}

V Embossment



Rule:
D (maximum) = 3T
R (minimum) = T for low carbon steel
R (minimum) = 2T for aluminum
R (minimum) = 2T for stainless steel

2.3 EDGE

INFORMATION

RULES:

1. The outermost condition of a sheet metal part is defined as an edge.
2. The most economical methods of producing an edge condition are by punching, shearing, or blanking.

REASONS: Punching, shearing and blanking are three basic standard manufacturing processes available for producing an edge condition. The tools used are:

- Hard dies
- Numerically controlled punch press
- Shearing

ALTERNATIVES: There are other more expensive, methods to produce an edge condition. Examples include, but are not limited to:

- Laser
- Chemical machining
- Electrical-discharge machining (EDM)
- Milling

ALTERNATIVE IMPACT: These other methods of producing edge conditions may require special equipment, which cannot produce adequate volumes in a cost-effective maner.

2.3.1 NOTCH

GUIDELINE

RULES:

1. An intrusion into the edge is called a notch. The minimum width of a notch is equal to the material thickness or 1.0 mm (0.04 inch), whichever is greater. See Figure 43, Notch.
2. The maximum recommended length of a notch is:

   straight or radius end — five times the width of the notch.
   V notch — two times the width of the notch.

NOTE

For special shapes, such as dovetail or keyhole, see a Producibility Engineer.

REASON: Punching a notch that is less than the material thickness or less than 1.00 mm (0.04 inch) wide or longer than the rule typically requires special tooling and/or processes that lead to increased tool maintenance and decreased tool life.

ALTERNATIVES: There are other methods of producing a notch narrower and/or deeper than the rule suggests. Examples include, but are not limited to:

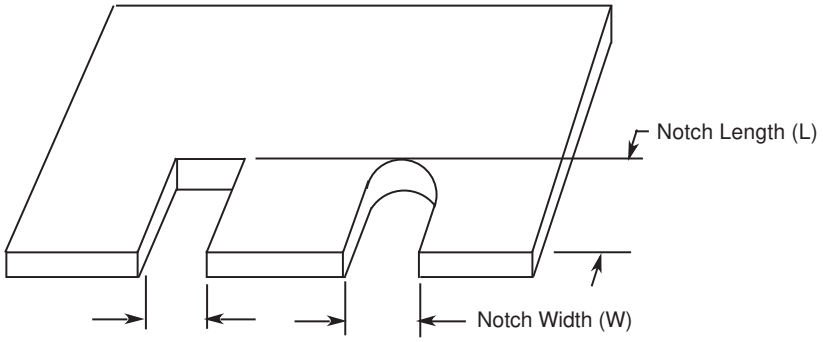
- Special designed tooling/dies
- Laser/milling
- Multiple punching (step and repeat)

ALTERNATIVE IMPACT: These other methods of producing a notch may require special tooling and/or secondary operations that lead to increased process and tooling costs.

Multiple punching (step and repeat) may cause burrs that may be *very* difficult to remove and may lead to manufacturing and quality problems.

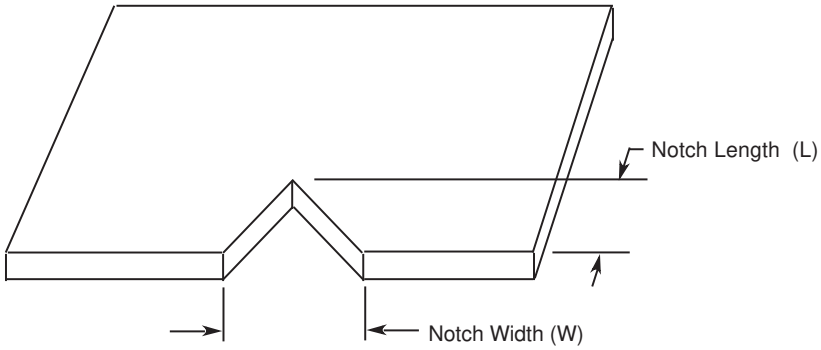
Figure 43: Notch

Notch



Rule: L (maximum) = 5W
W (minimum) = T or 1.0 mm (0.04 inch), whichever is larger.

V–Notch



Rule: L (maximum) = 2W

2.3.2 NOTCH TO HOLE

PRINCIPLE

RULE:

A minimum distance must be maintained between a hole and the edge of a notch. The minimum recommended distance between the hole and the notch is directly proportional to the size/shape of the hole feature and the material thickness. See Figure 44, Notch to Hole .

REASON: Holes punched too close to an edge may result in distortion of the hole and/or the web area between the hole and the edge, therefore creating a bulge. See Figure 44, Notch to Hole .

ALTERNATIVES: There are several alternative methods to modify a hole and/or edge to satisfy the rule (for example, make a hole into a notch or add a tab). See a Producibility Engineer for specific design recommendations.

There are other methods of producing holes closer to an edge than the rule suggests. Examples include, but are not limited to:

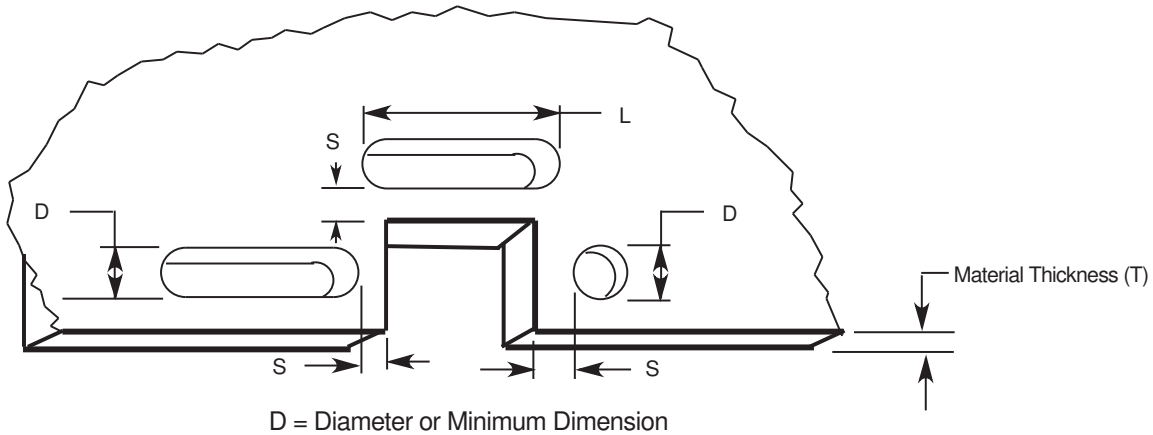
- Specially designed tooling/dies
- Milling
- Laser

If the design is not impaired by the edge distortion (bulge), a note can be added to the drawing/database stating that edge distortion is allowable (be specific on location).

ALTERNATIVE IMPACT: Positive or negative impacts of alternatives, have to be evaluated based upon specific design requirements. See a Producibility Engineer.

These other methods of producing holes require special tooling and/or secondary operations that lead to increased process and tooling costs.

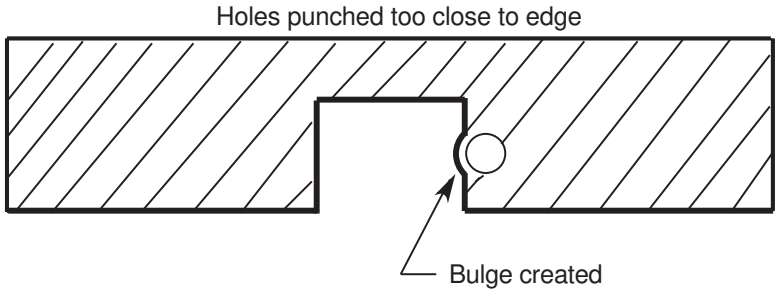
Figure 44: Notch to Hole



D = Diameter or Minimum Dimension
L = Length
T = Material Thickness
S = Minimum Dimension

Rule:

When D is 1T to 5T, then minimum S = 2.0 T
When D is 5T to 10T, then minimum S = 3.0 T
When D is greater than 10T (See Producibility Engineering for specific recommendations.)
When L is 1T to 10T, then minimum S = 2 T
When L is 10T to 25T, then minimum S = 4 T
When L is greater than 25T, then minimum S = 6 T (See Producibility Engineering for specific recommendations.)



2.3.3 NOTCH TO BEND

GUIDELINE

RULE:

The minimum distance from the inside surface of a bend to the edge of a notch is:

In a parallel plane — eight times the material thickness plus the radius of the bend.
In a perpendicular plane — three times the material thickness plus the radius of the bend.

See a Producibility Engineer for specific recommendations. See Figure 45, Notch to Bend.

REASON: Bending a flange closer to the edge than the rule suggests may lead to manufacturability (tooling and process) and/or quality (flange squareness and distortion) problems.

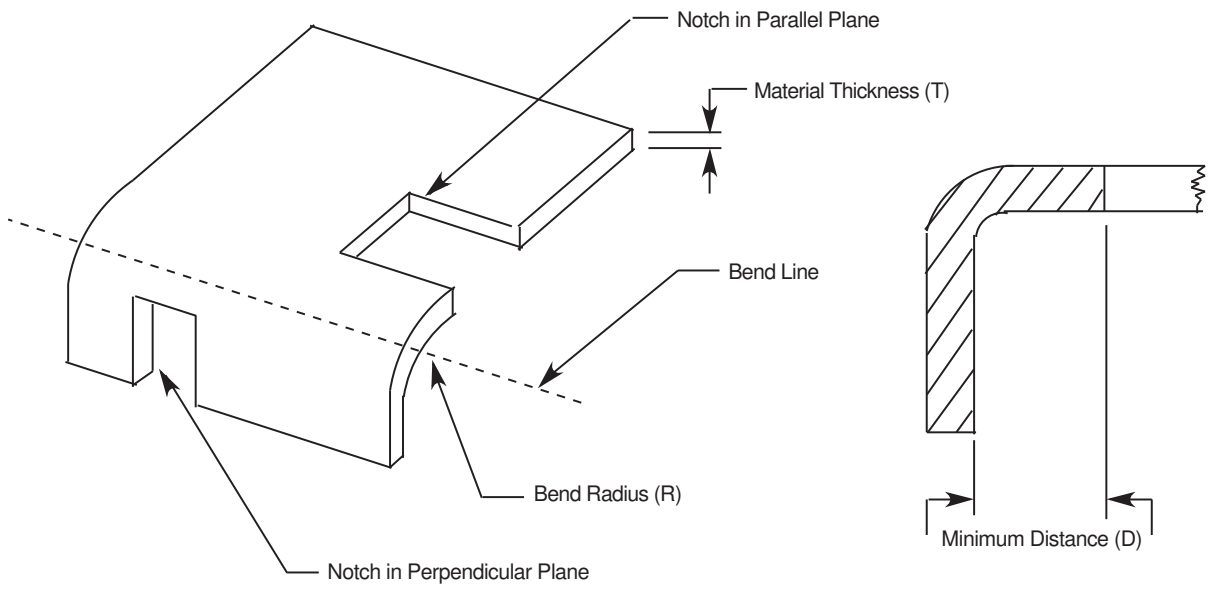
ALTERNATIVES: Notch through a bend.

There are other methods of producing a bent flange closer to a notch than the rule suggests. Examples include, but are not limited to:

- Specially designed tooling/dies
- Special processes (for example, punch notch after bending)

ALTERNATIVE IMPACT: These other methods of producing a bent flange close to a notch may require special tooling and/or secondary operations that lead to increased process and tooling costs.

Figure 45: Notch to Bend



For perpendicular plane notch: D (minimum) = 3T + R

For parallel plane notch: D (min) = 8T + R

2.3.4 NOTCH THROUGH A BEND

GUIDELINE

RULE:

When a notch through a bend is required, the bottom or side edges of the notch should extend a distance beyond the bend. The distance should be equal to the thickness of material plus the bend radius or two times the material thickness, whichever is greater. See Figure 46, Notch Through a Bend.

REASON: If a notch is made internal to a bend that will not extend beyond the bend (see rule), then the notch will distort or tear during the bending operation and this leads to manufacturing and/or quality problems.

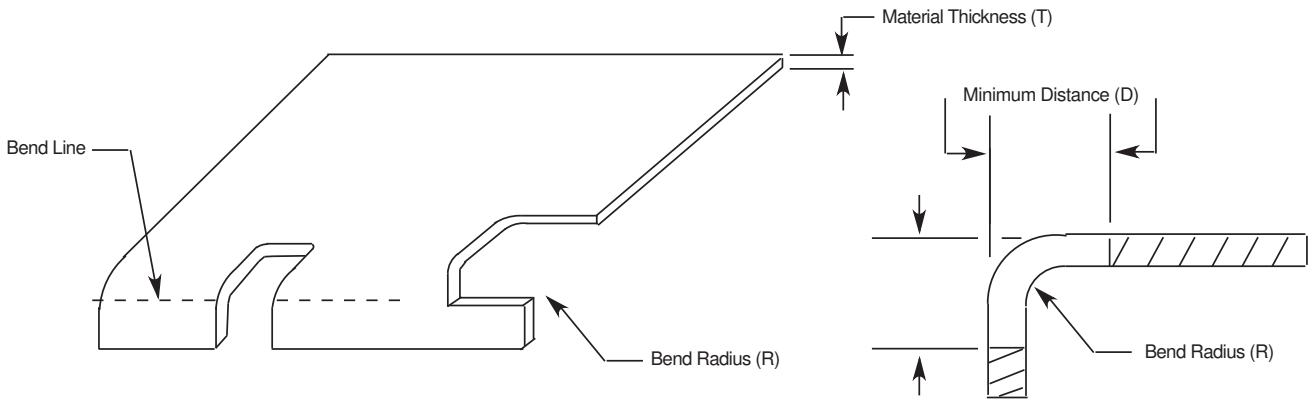
ALTERNATIVES: There are other methods of producing notches in sheet metal, internal to bends, and without distortion. Examples include, but are not limited to:

- Specially designed tooling/dies
- Machining after bending
  - Laser
  - Milling

If the design is not impaired by the notch distortion, a note can be added to the drawing/database stating that notch distortion is allowable (be specific on location).

ALTERNATIVE IMPACT: These other methods of producing notches may require special tooling and/or secondary operations that lead to increased process and tooling costs.

Figure 46: Notch Through a Bend



Note: Notch distortion is allowable.
Rule: D (minimum) = T + R or 2T, whichever is greater

2.3.5 NOTCH TO NOTCH

PRINCIPLE

RULE:

The recommended minimum distance between notches is three times the material thickness (thus creating a tab) or 3.20 mm (0.125 inch), whichever is greater. See Figure 47, Notch to Notch.

If you require several notches in a parallel plane, see a Producibility Engineer.

REASONS: Punching notches in sheet metal closer than the rule suggests may result in distortion of the notches.

Numerous notches punched too close together may result in distortion or warpage (oil-canning) of the material.

ALTERNATIVES: Combine the notches into a single wider notch.

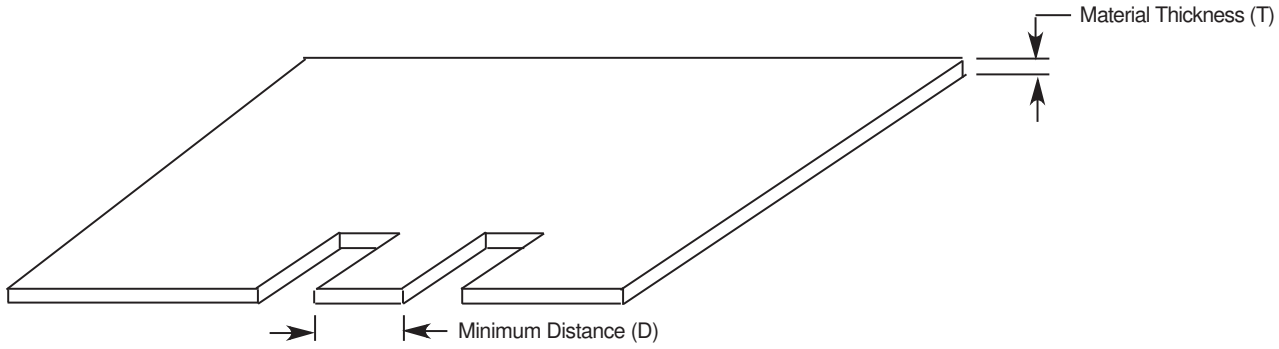
There are other methods of producing notches in sheet metal closer than the rule suggests. Examples include, but are not limited to:

- Specially designed tooling/dies
- Electrical-discharge machining (EDM)
- Laser

ALTERNATIVE IMPACT: Positive or negative impacts of the alternatives have to be evaluated based upon specific design requirements. See a Producibility Engineer.

These other methods of producing notches in sheet metal require special tooling and/or equipment that lead to increased process and tooling costs.

Figure 47: Notch to Notch



Rule: D (minimum) = 3T or 3.2 mm (0.125 inch), whichever is greater

2.3.6 TAB

GUIDELINE

RULE:

Protrusion from the edge is called a tab. The minimum width of a tab is equal to two times the material thickness or 3.20 mm (0.125 inch), whichever is greater. The maximum recommended length of a tab is five times the width of the tab. See a Producibility Engineer for specific design recommendations. See Figure 48, Tab.

REASONS: Punching a tab that is less than two times the material thickness or 3.20 mm (0.125 in) usually requires special tooling and/or processes that lead to increased tool maintenance and decreased tool life.

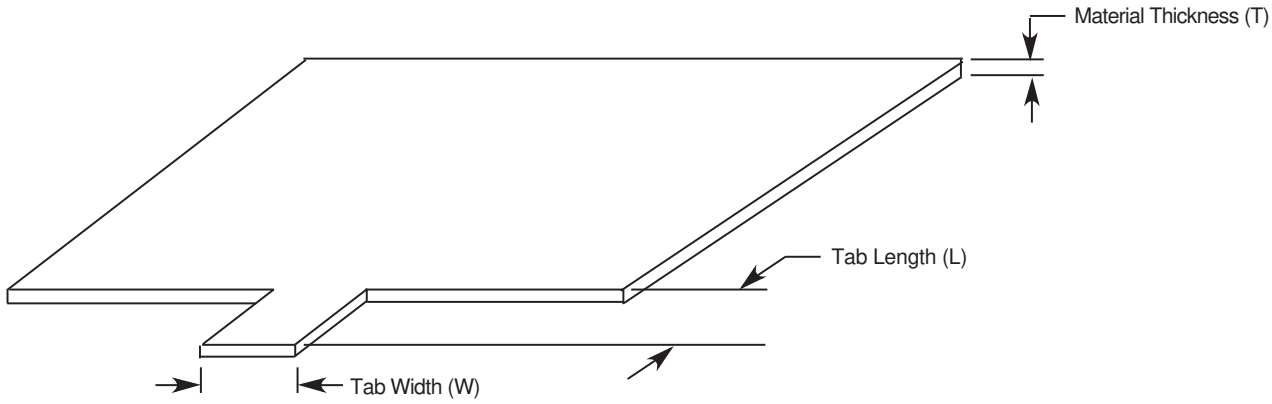
Punching a tab that is less than two times the material thickness or 3.20 mm (0.125 in) may cause the tab to deform (twist or tear).

ALTERNATIVES: There are other methods of producing a tab narrower and/or longer than the rule suggests. Examples include, but are not limited to:

- Special designed tooling/dies
- Laser
- Electrical-discharge machining (EDM)

ALTERNATIVE IMPACT: These other methods of producing a tab in sheet metal may require special tooling and/or secondary operations that lead to increased process and tooling costs.

Figure 48: Tab



Rule:

W (minimum) = 2T or 3.2 mm (0.125 inch), whichever is greater
L (maximum) = 5W

2.3.7 TAB TO TAB

PRINCIPLE

RULE:

The minimum recommended distance between tabs is equal to the material thickness or 1.00 mm (0.04 inch), whichever is greater (creating a notch).

If you require several tabs in a parallel plane, see a Producibility Engineer.

REASON: Punching tabs in sheet metal closer than the rule suggests may result in deformation (twisting or tearing) of the tabs.

ALTERNATIVES: Combine the tabs into a single wider tab.

Multiple tabs may be duplicated by the using additional fabricated and/or purchased parts (for example, purchased finger stock to be used in shielding).

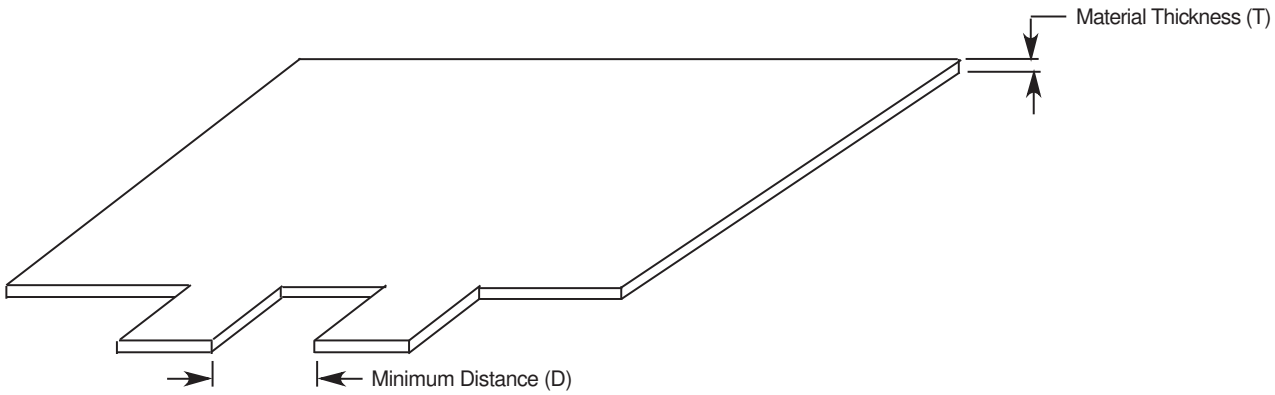
There are other methods of producing tabs in sheet metal closer than the rule suggests. Examples include, but are not limited to:

- Specially designed tooling/dies
- Laser
- Electrical-discharge machining (EDM)

ALTERNATIVE IMPACT: Positive or negative impacts of fabricated or purchased tab features have to be evaluated based upon specific design requirements. See a Producibility Engineer.

These other methods of producing tabs in sheet metal require special tooling and/or equipment that lead to increased process and tooling costs.

Figure 49: Tab to Tab



Rule:

D (minimum) = T or 1.0 mm (0.04 inch), whichever is greater

2.3.8 CORNER CONDITIONS

INFORMATION

RULES:

1. Corner conditions are designated as internal corners and external corners.

   Internal corners include:

   - Sharp (generally preferred)
   - Radius (preferred for hard tooling applications)

   External corners include:

   - Sharp (generally preferred)
   - Radius (preferred for hard tooling applications)
   - Chamfered (preferred for numerically controlled punch presses)

2. If, for tooling, safety, strength, or ease of assembly, the external corners have to be modified (broken), then the recommended note on the drawing/database can be as follows:

NOTE: All sharp corners should be broken. Radius or chamfer can be used interchangeably (be specific about location).

REASONS: Adding a radius or chamfer to a corner may increase the tooling and/or process cost and should not be specified unless specifically required for ease of tooling, safety, or assembly.

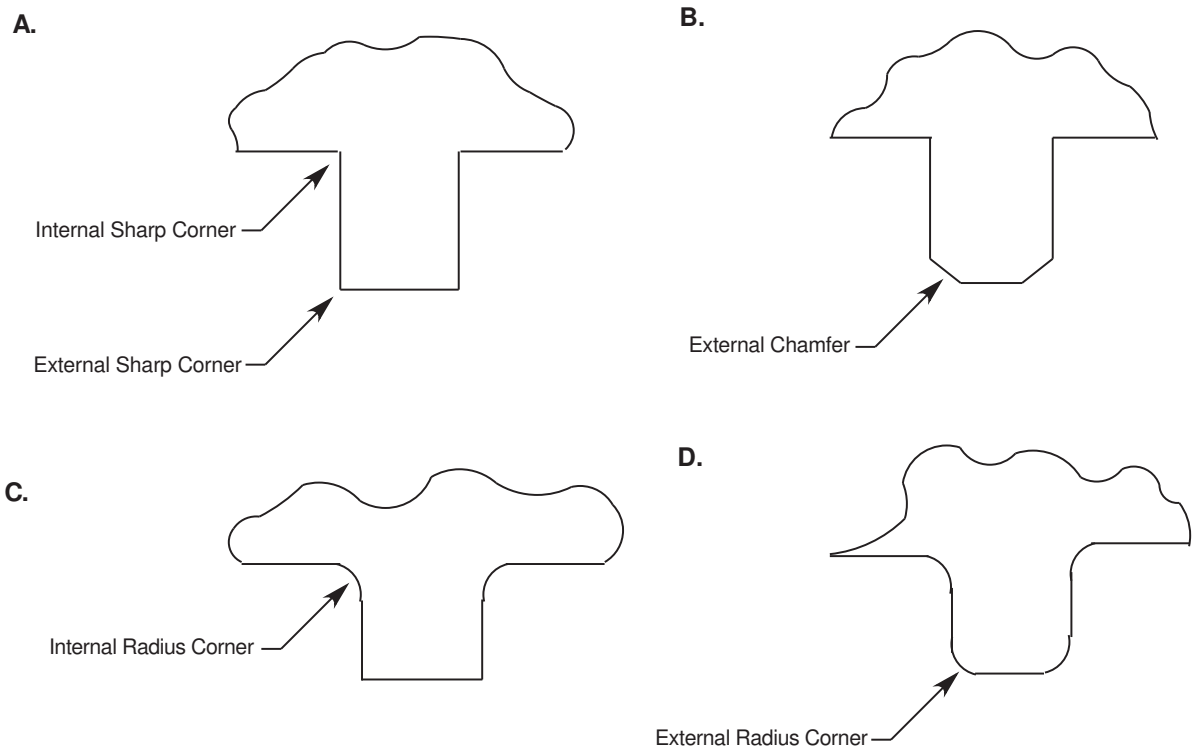
Corner conditions are produced in various shapes to provide flexibility in meeting design and/or manufacturing requirements.

ALTERNATIVES: Corner conditions may be simulated by using additional operations or purchased parts. Examples include, but are not limited to:

- Purchased components (grommets)
- Formed features (hem)

ALTERNATIVE IMPACT: Positive or negative impacts of a fabricated or purchased corner condition feature have to be evaluated based upon specific design requirements. See a Producibility Engineer.

Figure 50: Corner Conditions



Note:

Sharp corners may be broken by a radius or chamfer and can be used interchangeably.

2.3.9 CORNER CONDITION - RADIUS

PRINCIPLE

RULES:

1. The minimum recommended radius to be used as a corner condition is two times the material thickness, or 1.50 mm (0.06 inch), whichever is greater. See Figure 51, Corner Condition - Radius.
2. A radius is used for either an internal or external corner condition. See Figure 51, Corner Condition - Radius.
3. Unless otherwise specified, a radius or chamfer can be used interchangeably, with the following recommended note on the drawing/database (be specific on location).

   Note: All sharp corners to be broken by means of a radius or chamfer.

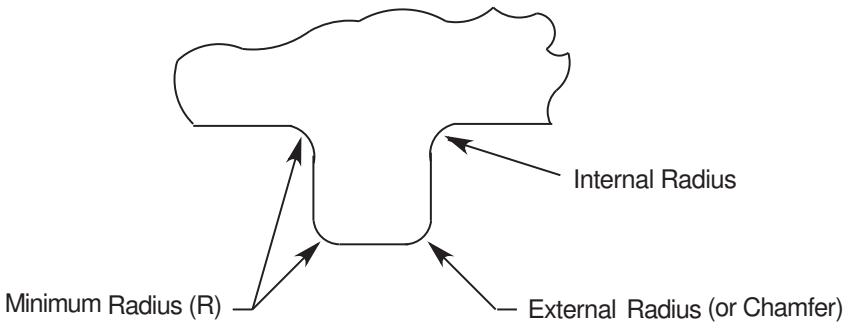
REASON: Specifying a radius which is less than two times the material thickness or less than 1.50 mm (0.06 inch) will not improve conditions of safety.

ALTERNATIVES: Use a chamfer.

Leave corners sharp.

ALTERNATIVE IMPACT: Positive or negative impacts of alternative corner conditions have to be evaluated based upon specific design and/or manufacturing requirements. See a Producibility Engineer.

Figure 51: Corner Condition - Radius



Note:

Sharp corners may be broken by a radius or chamfer for external corners only.

Rule:

T = material thickness

R (minimum) = 2 T or 1.5 mm (0.06 in), whichever is greater.

2.3.10 CORNER CONDITION - CHAMFER

PRINCIPLE

RULES:

1. The minimum recommended chamfer used as a corner condition is two times the material thickness or 1.50 mm (0.06 inch), whichever is greater, at a recommended angle of forty-five degrees. See Figure 52, Corner Condition - Chamfer.
2. A chamfer is recommended for use only on external corners. See Figure 52, Corner Condition - Chamfer.
3. Unless otherwise specified, a radius or chamfer should be used interchangeably with the following recommended note on the drawing/database (be specific on location).

   NOTE: All sharp corners to be broken by means of a radius or chamfer.

REASONS: A chamfer which is less than two times the material thickness or less than 1.50 mm (0.06 inch) leads to increased (hard) tooling maintenance and decreased tool life.

A chamfer which is at an angle other than forty-five degrees leads to increased operational and tooling costs.

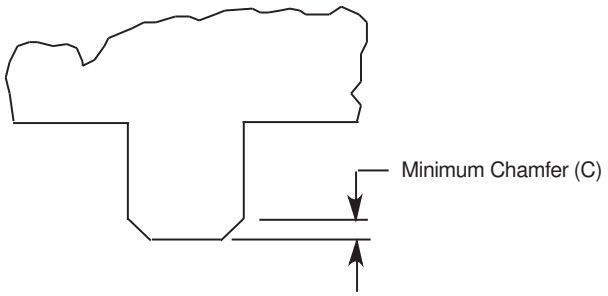
Using a chamfer as an internal corner condition may lead to increased operational and/or tooling costs.

ALTERNATIVES: Use a Radius.

Leave corners sharp.

ALTERNATIVE IMPACT: Positive or negative impacts of alternative corner conditions have to be evaluated based upon specific design and/or manufacturing requirements. See a Producibility Engineer.

Figure 52: Corner Condition - Chamfer



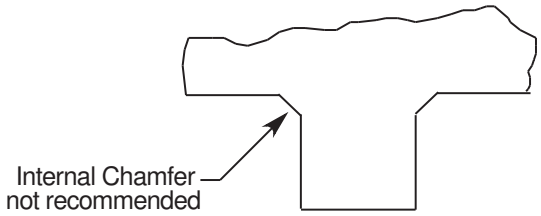
Note:

Sharp corners may be broken by a radius or chamfer.

Rule:

T = Material thickness

C (minimum) = 2 T or 1.5 mm (0.06 in), whichever is greater.



2.4 SPECIAL FEATURES

2.4.1 EXTRUDED HOLE

INFORMATION

RULES:

1. Piercing and extruding a hole in a single operation in sheet metal may be accomplished by using specially designed tooling. See Figure 53, Special Features (Extruded Hole).
2. An extruded hole, based upon the process, is specific for roll-form threading applications and is **not recommended for the cut-tap threading.**

NOTE

Extruded holes are not recommended for aluminum or stainless steel due to fracturing. See a Producibility Engineer.

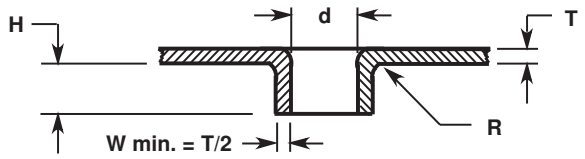
REASON: The process is a cost-effective method of producing a roll-form threaded hole in sheet metal.

ALTERNATIVES: There are other methods of producing threaded holes in sheet metal. Examples include, but are not limited to:

- Use of threaded inserts
- Use of heavier material (and cut-tap)

ALTERNATIVE IMPACT: Positive or negative impacts of the alternatives have to be evaluated based upon specific design requirements. See a Producibility Engineer.

Figure 53: Special Features (Extruded Hole)



d = hole diameter
T = material thickness
R = extruded radius
H = height of extruded hole
W = minimum wall thickness

2.4.2 EXTRUDED HOLE - ROLL FORM THREADING

PRINCIPLE

RULES:

1. When piercing and extruding a hole in sheet metal to be used for roll-form threading, the material thickness should be a minimum one-fifth, to a maximum of three-quarters, of the diameter of the thread. See Figure 54, Extruded Hole - Roll Form Threading.
2. The minimum extrusion wall thickness should not be less than one-half the material thickness.

NOTE

Extruded holes are not recommended for aluminum or stainless steel due to fracturing. See a Producibility Engineer.

REASON: Specifying an extruded hole for roll-form threading that conforms to the rule will provide enough strength to develop a reasonable proportion of the strength of the screw.

ALTERNATIVES: There are other methods of producing a threaded hole in sheet metal. Examples include, but are not limited to:

- Use of threaded inserts
- Use of heavier material (and cut-tap)

ALTERNATIVE IMPACT: Positive or negative impacts of the alternatives have to be evaluated based upon specific design requirements. See a Producibility Engineer.

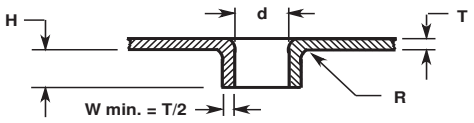
Figure 54: Extruded Hole - Roll Form Threading

RECOMMENDED EXTRUSION PILOT HOLE DIAMETERS (d) FOR ROLL FORMED THREADS

Material – Steel Sheet – Low Carbon Grades – Average ROCKWELL "B" 55–75

SCREW SIZE	APPROXIMATE MATERIAL THICKNESS T											
	.02	.03	.04	.06	.09	.13	.16	.19	.22	.25	.31	.38
2–56	.075/.076	.075/.077	.076/.078									
3–48	.086/.087	.086/.088	.087/.089									
4–40	.096/.097	.096/.098	.097/.099									
5–40	.109/.110	.109/.111	.110/.112	.111/.113				HOLE SIZES ARE BASED ON PERCENT THREAD ENGAGEMENT FOR OPTIMUM PERFORMANCE				
6–32	.118/.120	.118/.121	.119/.122	.120/.123	.122/.125							
8–32	.144/.146	.144/.147	.145/.148	.146/.149	.147/.150	.148/.152						
10–24	.163/.165	.163/.166	.164/.167	.165/.168	.166/.170	.168/.173						
10–32	.170/.172	.170/.173	.171/.174	.172/.175	.173/.176	.174/.177						
12–24	.189/.191	.189/.192	.190/.193	.191/.194	.192/.196	.193/.197	.195/.200	.198/.203				
1/4–20			.218/.220	.218/.221	.219/.223	.221/.225	.224/.228	.227/.231	.229/.233	.231/.235		
5/16–18				.277/.279	.278/.280	.279/.281	.280/.283	.281/.285	.283/.288	.285/.290		
3/8–16						.335/.337	.336/.338	.337/.340	.339/.343	.342/.346	.345/.349	
7/16–14							.392/.395	.394/.397	.396/.400	.398/.402	.401/.405	.404/.409
1/2–13								.450/.453	.452/.455	.454/.457	.456/.460	.460/.464

FLANGE DIMENSIONS



HOLE DIA. d	APPROXIMATE MATERIALTHICKNESS T															
	.024 – .035		.042 – .048		.060		.075		.090		.106		.118		.122	
	$H^{+.040}_{-.000}$	$R^{+.005}_{-.000}$	$H^{+.040}_{-.000}$	$R^{+.005}_{-.000}$	$H^{+.040}_{-.000}$	$R^{+.005}_{-.000}$	$H^{+.040}_{-.000}$	$R^{+.005}_{-.000}$	$H^{+.040}_{-.000}$	$R^{+.005}_{-.000}$	$H^{+.040}_{-.000}$	$R^{+.005}_{-.000}$	$H^{+.040}_{-.000}$	$R^{+.005}_{-.000}$	$H^{+.040}_{-.000}$	$R^{+.005}_{-.000}$
Over .08 Thru .10	.040	.005	.040	.005	.040	.006	.043	.008	.043	.010						
Over .10 Thru .13	.047	.005	.047	.005	.047	.006	.052	.008	.052	.010	.054	.010				
Over .13 Thru .15	.053	.005	.053	.005	.053	.006	.060	.008	.060	.010	.063	.010	.063	.012	.072	.012
Over .15 Thru .18			.060	.005	.061	.006	.070	.008	.070	.010	.075	.010	.075	.012	.087	.013
Over .18 Thru .22			.070	.005	.070	.006	.090	.008	.090	.010	.095	.010	.095	.012	.104	.013
Over .22 Thru .26					.075	.006	.100	.008	.100	.010	.105	.010	.105	.012	.120	.013
Over .26 Thru .30					.083	.006	.116	.008	.116	.010	.125	.010	.125	.012	.140	.013
Over .30 Thru .34							.130	.008	.130	.010	.140	.010	.140	.012	.154	.013
Over .34 Thru .38									.140	.010	.155	.010	.155	.012	.170	.013
Over .38 Thru .43									.150	.010	.170	.010	.170	.012	.184	.013

NOTE: Table example covers the normal range of material thickness for practal single punch extrusion. Minimum extrusion wall thickness (W) should never be less than one–half material thickness (T).

2.4.3 EXTRUDED HOLE TO EXTRUDED HOLE

PRINCIPLE

RULE:

The minimum recommended distance between two extruded holes (from the major diameter of each extrusion) is six times the material thickness. See Figure 55, Extruded Hole to Extruded Hole.

NOTE

Extruded holes are not recommended for aluminum or stainless steel due to fracturing. See a Producibility Engineer.

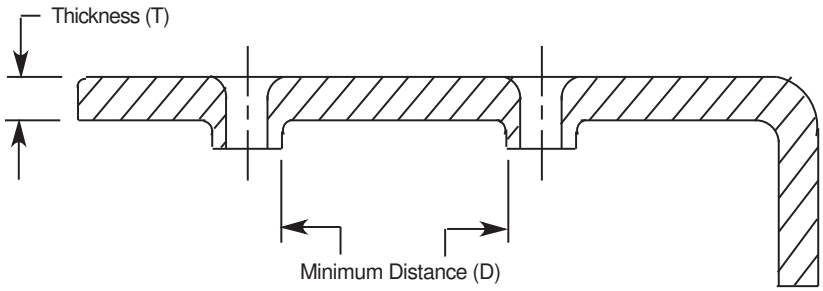
REASON: Extruded holes punched too close together may result in distortion of the holes and/or the web area between them.

ALTERNATIVES: There are other methods of producing extruded holes in sheet metal closer than the rule suggests. Examples include, but are not limited to:

- Specially designed tooling/dies
- Use of multiple parts and secondary operations (for example, various assembly techniques)

ALTERNATIVE IMPACT: These other methods of producing extruded hole features in sheet metal require special tooling and/or secondary operations that lead to increased process and tooling costs.

Figure 55: Extruded Hole to Extruded Hole



Rule:
D (minimum) = 6T

2.4.4 EXTRUDED HOLE TO EDGE

PRINCIPLE

RULE:

The minimum recommended distance from the major diameter of the extrusion in an extruded hole to an edge is three times the material thickness. See Figure 56, Extruded Hole to Edge.

NOTE

Extruded holes are not recommended for aluminum or stainless steel due to fracturing. See a Producibility Engineer.

REASON: Extruded holes punched too close to an edge may result in distortion of the hole and/or web area between the hole and the edge (creating a bulge).

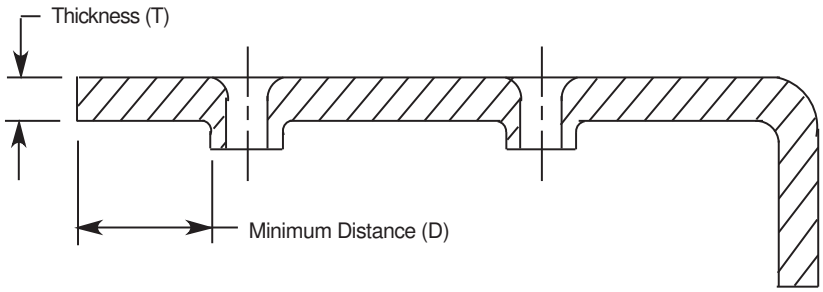
ALTERNATIVES: Extend the edge local to the hole to satisfy the rule (for example, add a tab).

There are other methods of producing extruded holes closer to an edge than the rule suggests. Examples include, but are not limited to:

- Specially designed tooling/dies
- Machining after extruding

ALTERNATIVE IMPACT: These other methods of producing extruded holes in sheet metal require special tooling and/or secondary operations that lead to increased process and tooling costs.

Figure 56: Extruded Hole to Edge



Rule:
D (minimum) = 3T

2.4.5 EXTRUDED HOLE TO BEND

PRINCIPLE

RULE:

The minimum recommended distance from the major diameter of an extruded hole to a bend is three times the material thickness plus the bend radius. See Figure 57, Extruded Hole to Bend.

NOTE

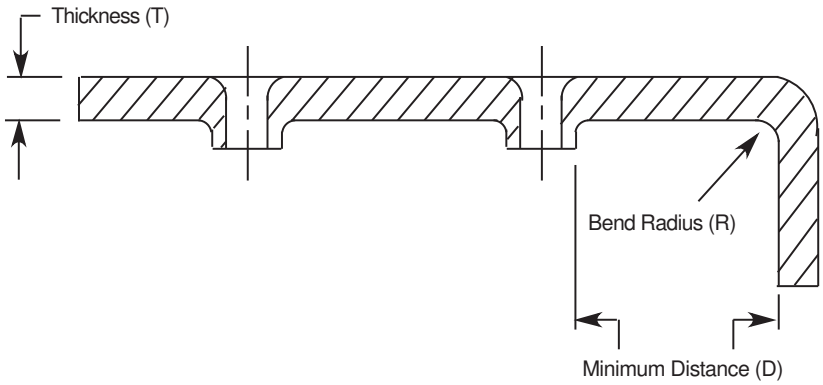
Extruded holes are not recommended for aluminum or stainless steel due to fracturing. See a Producibility Engineer.

REASON: Extruded holes positioned too close to a bend may result in hole distortion.

ALTERNATIVES: Extruded holes may also be located much closer to a bend than the rule suggests by using relief cuts/slots (for example, add a tab).

ALTERNATIVE IMPACT: Design considerations and/or constraints may impair the adding of bend reliefs. They will also make it necessary to incur (minimal) operational and/or tooling cost increases.

Figure 57: Extruded Hole to Bend



Rule:

D (minimum) = 3T + R

2.4.6 LANCE - OPEN

GUIDELINE

RULE:

An open lance is the cutting (shearing) and forming of the material in one operation with no relief cuts. For low carbon steel, the minimum width is two times the material thickness or 3.20 mm (0.125 inch), whichever is greater, with a maximum length of five times the width. For aluminum or stainless steel, the minimum width should be three times the material thickness or 4 mm (0.16 inch), whichever is greater, with a maximum length of five times the width.

See a Producibility Engineer for specific design recommendations. See Figure 58, Lance - Open.

NOTE

A sheared lance is difficult to deburr.

REASONS: Specifying an open lance that does not conform to the rule may cause distortion or tearing of the lance.

Shear and form is the strongest and most economical method of producing a lance.

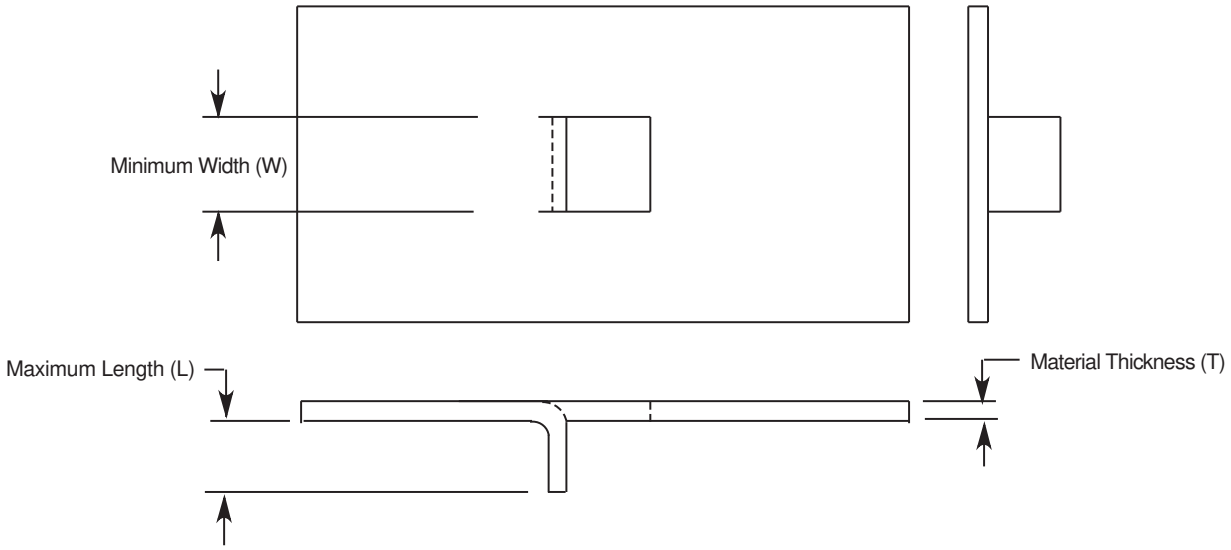
ALTERNATIVES: A feature similar to a lance may be produced by making relief cuts/slots first, and then form as a secondary operation. See Figure 58, Lance - Open.

There are other methods of producing a lance feature, such as in the use of multiple parts and and assembly techniques (for example, welding).

ALTERNATIVE IMPACT: The use of relief cuts/slots require the use of secondary operations that lead to increased process and/or tooling costs.

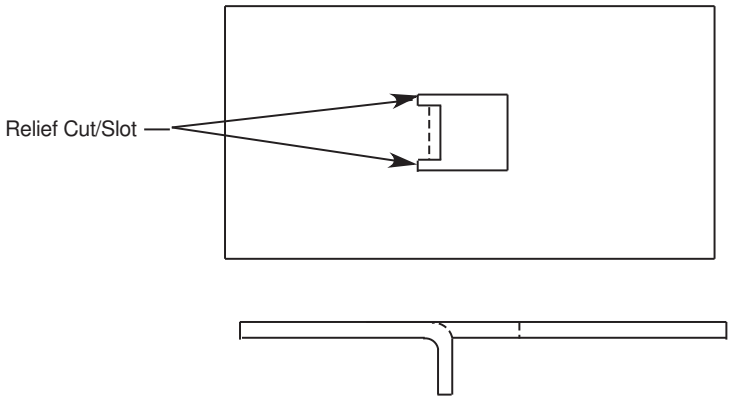
Positive or negative impacts of an assembly (for example, welded construction) have to be evaluated based upon specific design requirements. See a Producibility Engineer.

Figure 58: Lance - Open



Rule: W (minimum) = 2T or 3.2 mm (0.125 of an inch), whichever is greater for low carbon steel.
W (minimum) = 3T or 4 mm (0.16 inch) for aluminum
W (minimum) = 3T or 4 mm (0.16 inch) for stainless steel
L (maximum) = 5 W

Alternative:



2.4.7 LANCE - CLOSED

GUIDELINE

RULE:

A closed lance is attached on two sides. It is created by the cutting (shearing) and forming of the material in one operation with no relief cuts.

For low carbon steel, the minimum width is two times the material thickness or 1.50 mm (0.06 inch), whichever is greater to a maximum height of five times the material thickness at a 45-degree angle (or full radius), with no constraints on the length.

For aluminum or stainless steel the minimum width is three times the material thickness or 2.50 mm (0.100 inch), whichever is greater to a maximum height of three times the material thickness at a 45-degree angle (or full radius), with no constraints on the length.

See a Producibility Engineer for specific design recommendations. See Figure 59, Lance - Closed.

NOTE

A sheared lance is *very* difficult to deburr.

REASONS: Specifying a closed lance that does not conform to the rule may cause distortion and/or tearing of the lance.

Shear and form is the strongest and most economical method of producing a lance.

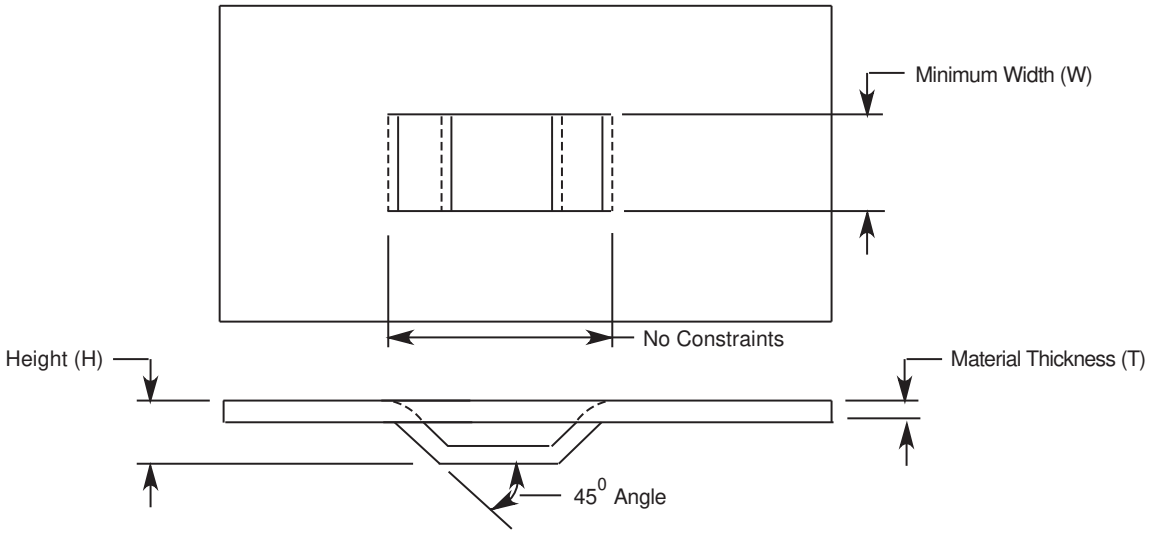
ALTERNATIVES: A similar feature to a lance may be produced by providing relief cuts/slots first and then form as a secondary operation. See Figure 59, Lance - Closed.

There are other methods of producing a lance feature, such as using multiple parts and and assembly techniques (for example, welding).

ALTERNATIVE IMPACT: The use of relief cuts/slots require the use of secondary operations that lead to increased process and/or tooling costs.

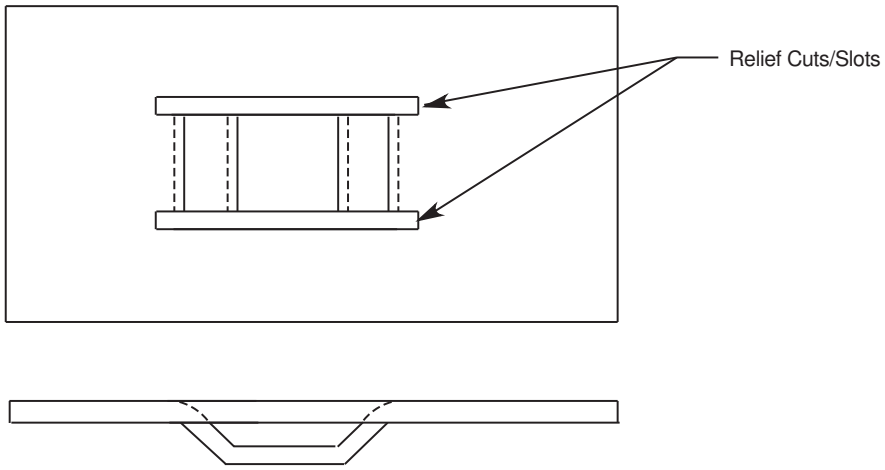
Positive or negative impacts of an assembly (for example, welded construction) have to be evaluated based upon specific design requirements. See a Producibility Engineer.

Figure 59: Lance - Closed



Rule: W (minimum) = 2T or 1.5 mm (0.06 of an inch) whichever is greater for low carbon steel.
W (minimum) = 3T or 2.5 mm (0.10 inch) for aluminum or stainless steel
H (maximum) = 5T for low carbon steel
H (maximum) = 3T for aluminum or stainless steel

Alternative:



2.4.8 LANCE TO EDGE

GUIDELINE

RULE:

See a Producibility Engineer for recommendations specific to design requirements.

REASON: There are too many variations to specify a general rule for minimum distance of a lance to an edge.

ALTERNATIVE: N/A

ALTERNATIVE IMPACT: N/A

2.4.9 LANCE TO BEND

GUIDELINE

RULE:

The minimum distance from the inside surface of the bend to the edge of a lance is:

In a parallel plane — eight times the material thickness plus the radius of the bend. See Figure 60, Lance to Bend.
In a perpendicular plane — ten times the material thickness plus the radius of the bend.
See a Producibility Engineer for specific recommendations. See Figure 60, Lance to Bend.

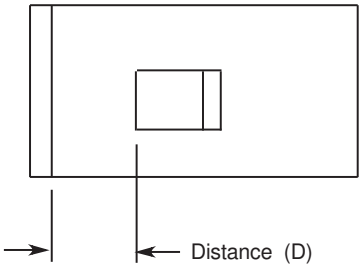
REASON: Bending a flange closer to a lance than the rule suggests may lead to manufacturability (tooling and process) and/or quality (distortion) problems.

ALTERNATIVES: There are other methods of producing a lance in sheet metal closer to a bent flange than the rule suggests. Examples include, but are not limited to:

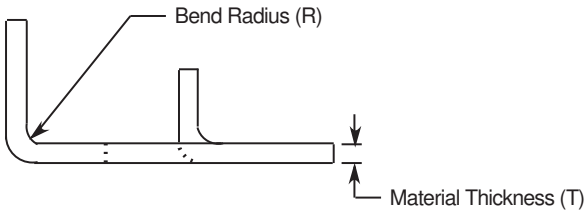
- Specially designed tooling/dies
- Special process (for example, lance after bending)

ALTERNATIVE IMPACT: These other methods of producing a bent flange close to a lance may require special tooling and/or secondary operations that lead to increased process and tooling costs.

Figure 60: Lance to Bend

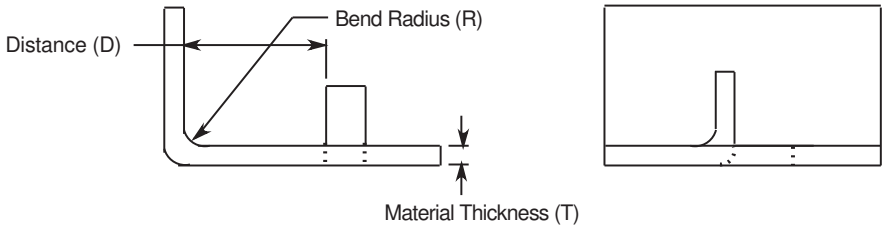


Parallel Plane:



Rule: D (minimum) = 8 T + R

Perpendicular Plane:



Rule: D (minimum) = 10 T + R

2.4.10 LANCE TO HOLE

GUIDELINE

RULE:

The minimum distance from a hole to a lance is three times the material thickness. See a Producibility Engineer for specific design recommendations. See Figure 61, Lance to Hole.

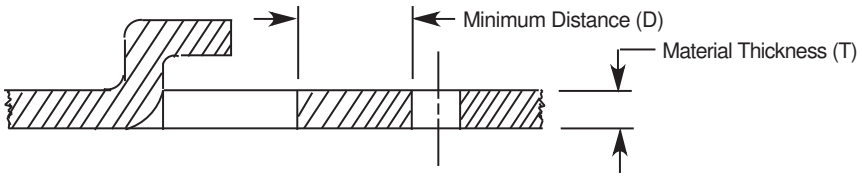
REASON: A hole punched too close to a lance may result in distortion of the hole and/or lance.

ALTERNATIVES: There are other methods of producing holes closer to a lance than the rule suggests. Examples include, but are not limited to:

- Specially designed tooling/dies
- Special processes (for example, punch hole after lance)
- Other processes (for example, laser, chemical etching)

ALTERNATIVE IMPACT: These other methods of producing holes close to a lance may require special tooling, processes and/or secondary operations that lead to increased process and tooling costs.

Figure 61: Lance to Hole



Rule: D (minimum) = 3 T

2.4.11 LANCE TO LANCE

GUIDELINE

RULE:

See a Producibility Engineer for recommendations specific to design requirements.

REASON: There are too many variations to specify a general rule for minimum distance between lances.

ALTERNATIVE: N/A

ALTERNATIVE IMPACT: N/A

2.4.12 LOUVERS

INFORMATION

RULE:

A louver is a special type of lance and is typically used for permitting air flow. See Figure 62, Louvers.

NOTE

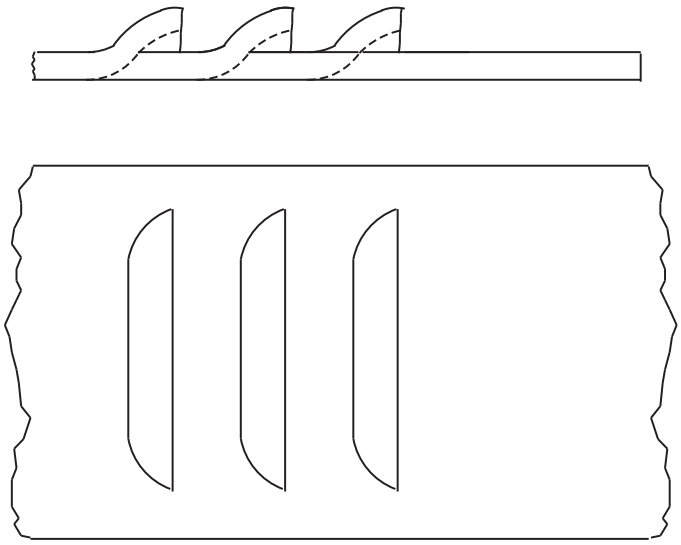
It should be understood that due to the potential variation in size and shape of louvers, general rules *cannot* be created. See a Producibility Engineer for recommendations.

REASON: Louvers are not limited to a particular shape or size which makes them difficult to define. Specific design requirements should be reviewed with a producibility engineer for manufacturing considerations.

ALTERNATIVE: Modify the design by using multiple parts. For example, provide a cutout and attach (see subhead 2.6 Joining Features) a separate, specially perforated (metal or plastic) insert.

ALTERNATIVE IMPACT: Positive or negative impacts of a multiple part fabrication have to be evaluated based upon specific design requirements. See a Producibility Engineer.

Figure 62: Louvers



2.4.13 COUNTERSINK

INFORMATION

RULE:

A countersink is a conical depression typically used in conjunction with assembly hardware, such as flat head rivets and/or flat head screws.

Countersinks are designated as formed and coined. See Figure 63, Countersink.

REASON: Formed and coined countersinks may be produced by standard sheet metal manufacturing processes.

ALTERNATIVES: There are other, more expensive, methods of producing countersink features in sheet metal, such as in the process of metal cutting.

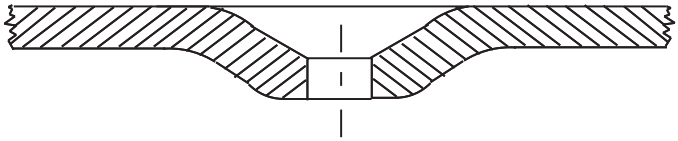
A counterbore, along with alternate assembly hardware, may be used.

ALTERNATIVE IMPACT: These other methods of producing countersinks in sheet metal require additional and more costly processes (secondary operations).

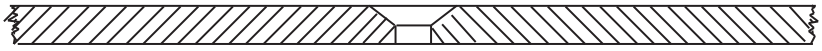
Positive or negative impacts of using a countersink versus a counterbore have to be evaluated based upon specific design requirements. See a Producibility Engineer.

Figure 63: Countersink

A. Formed:



B. Coined:



2.4.14 COUNTERSINK - FORMED

GUIDELINE

RULES:

1. A formed countersink is a conical depression. It has a maximum depth equal to 3½ times the material thickness with an included angle equal to the head angle of the hardware (for example, 82°, 100°). See Figure 64, Countersink - Formed.
2. A minimum of 50% contact is required between the hardware (head) and the countersink.

NOTE

A formed countersink usually results in pilot hole distortion requiring additional tolerance consideration.

REASONS: Forming a countersink in sheet metal deeper than the rule specifies may cause material fracturing and/or hole distortion leading to manufacturing and/or quality problems.

A minimum of 50% surface contact is required between the countersink and hardware to insure the proper use and fit of the hardware.

ALTERNATIVES: There are other methods of producing a countersink in sheet metal. Examples include, but are not limited to:

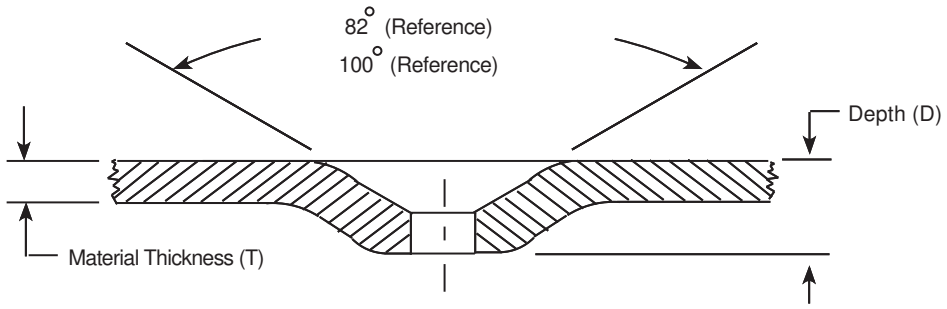
- Coining
- Machining

If the design is not impaired by countersink hole distortion, a note can be added to the drawing/database stating that hole distortion is allowable (be specific on location).

ALTERNATIVE IMPACT: Positive or negative impacts of the alternatives have to be evaluated based upon specific design requirements. See a Producibility Engineer.

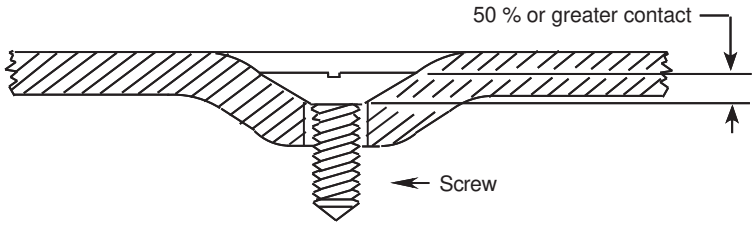
Figure 64: Countersink - Formed

A. Countersink – Formed



Rule: D (maximum) = 3 .5 T

B. Head to Countersink Contact



Rule: 50 % or greater contact

2.4.15 COUNTERSINK - COINED

GUIDELINE

RULES:

1. A coined countersink is a conical depression. It has a maximum depth equal to 80% of the material thickness with an included angle equal to the head angle of the hardware (for example, 82°, 100°).
2. A minimum of 50% contact is required between the hardware (head) and the countersink.

   (See Figure 65, Countersink - Coined).

NOTE

A coined countersink usually results in surface bulge and pilot hole distortion requiring additional tolerance consideration. See Figure 65, Countersink - Coined.

Coined countersinks are not recommended for stainless steel.

REASONS: Coining a countersink in sheet metal deeper than the rule specifies may cause excessive material bulging and hole distortion that leads to manufacturing and/or quality problems.

A minimum of 50% surface contact is required between the countersink and hardware to ensure the proper use and fit of the hardware.

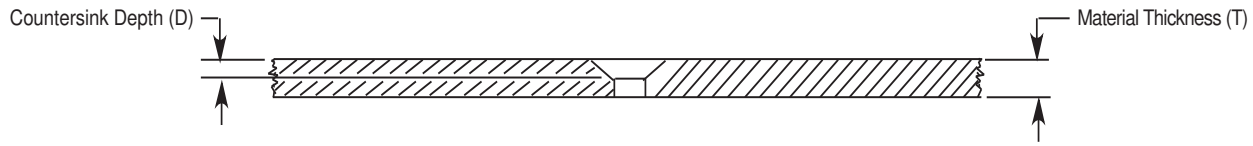
ALTERNATIVES: There are other methods of producing a deeper countersink in sheet metal. Examples include, but are not limited to:

- Forming
- Machining

If the design is not impaired by countersink material bulging, a note can be added to the drawing/database stating material bulging is allowable (be specific on location).

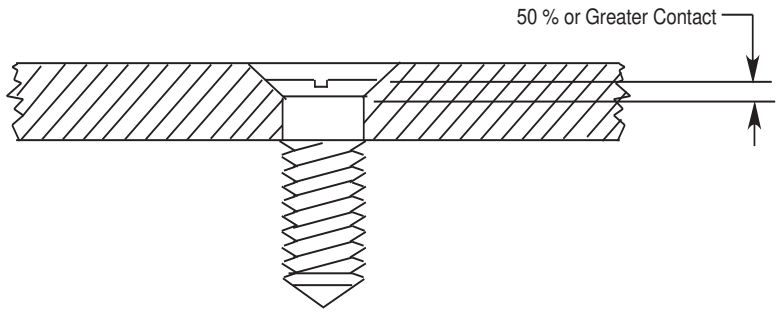
ALTERNATIVE IMPACT: Positive or negative impacts of the alternatives have to be evaluated based upon specific design requirements. See a Producibility Engineer.

Figure 65: Countersink - Coined



Rule: " D" (maximum) = 80 % T

Not Recommended for stainless steel



2.4.16 COUNTERSINK TO COUNTERSINK

PRINCIPLE

RULE:

The minimum recommended distance between the major diameter of two countersinks is eight times the material thickness. See Figure 66, Countersink to Countersink.

NOTE

Coined countersinks are not recommended for stainless steel.

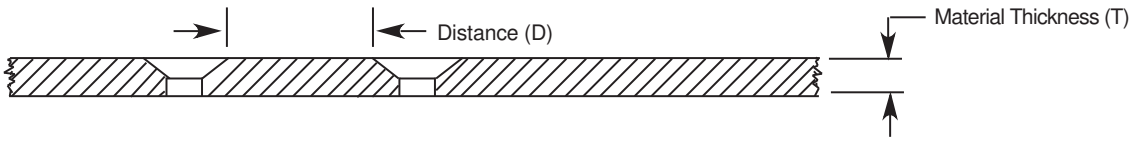
REASON: Countersinks formed too close together may result in distortion of the countersinks and/or web area between them.

ALTERNATIVES: There are other methods of producing countersinks in sheet metal closer than the rule suggests. Examples include, but are not limited to:

- Specially designed tooling/dies
- Machined countersinks

ALTERNATIVE IMPACT: These other methods of producing countersinks in sheet metal require special tooling and/or secondary operations that lead to increased process and tooling costs.

Figure 66: Countersink to Countersink



Rule: D (minimum) = 8 T

Coined countersinks are not recommended for stainless steel .

2.4.17 COUNTERSINK TO EDGE

PRINCIPLE

RULE:

The minimum recommended distance from the major diameter of a countersink to an edge is four times the material thickness. See Figure 67, Countersink to Edge.

NOTE

Coined countersinks are not recommended for stainless steel.

REASON: Countersinks formed too close to an edge may result in distortion of the countersink and/or web area between the countersink and the edge (creating a bulge) leading to manufacturing and/or quality problems.

ALTERNATIVES: Extend the edge local to the countersink to satisfy the rule.

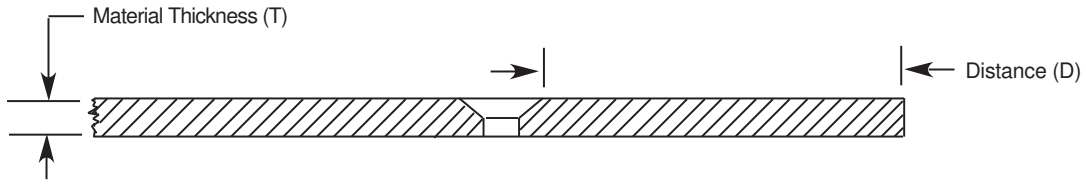
There are other methods of producing countersinks closer to an edge than the rule suggests. Examples include, but are not limited to:

- Specially designed tooling/dies
- Machining after forming/coining

If the design is not impaired by the edge distortion (bulge), a note can be added to the drawing/database stating that edge distortion is allowable (be specific on location).

ALTERNATIVE IMPACT: These other methods of producing countersinks in sheet metal require special tooling and/or secondary operations that lead to increased process and tooling costs.

Figure 67: Countersink to Edge



Rule: D (minimum) = 4 T

Coined countersinks are not recommended for stainless steel .

2.4.18 COUNTERSINK TO BEND

PRINCIPLE

RULE:

The minimum recommended distance from the major diameter of a countersink to a bend is four times the material thickness plus the bend radius. See Figure 68, Countersink to Bend.

NOTE

Coined countersinks are not recommended for stainless steel.

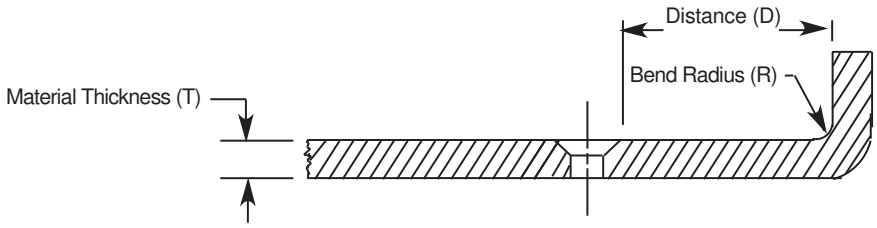
REASON: Countersinks positioned too close to a bend may result in distortion of the countersink leading to manufacturing and/or quality problems.

ALTERNATIVES: Countersinks may be located much closer to a bend than the rule suggests by using relief cuts/slots, effectively adding a tab.

If the design is not impaired by the edge distortion (bulge), a note can be added to the drawing/database stating that edge distortion is allowable (be specific on location).

ALTERNATIVE IMPACT: Design considerations and/or constraints may impair the adding of bend reliefs. They will also make it necessary to incur (minimal) operational and/or tooling cost increases.

Figure 68: Countersink to Bend



Rule: D (minimum) = 4 T + R

Coined countersinks are not recommended for stainless steel .

2.4.19 COUNTERBORE

INFORMATION

RULE:

A counterbore is a stepped hole coined into the surface of the material and typically used in conjunction with assembly hardware. See Figure 69, Counterbore.

NOTE

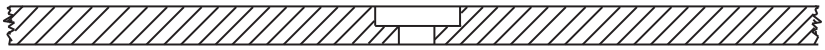
Coined counterbores are not recommended for stainless steel.

REASON: A counterbore may be produced by means of standard sheet metal manufacturing processes, such as punching and coining.

ALTERNATIVES: There are other, usually more expensive, methods of producing counterbore features in sheet metal, such as in the process of metal cutting.

ALTERNATIVE IMPACT: These other methods of producing a counterbore in sheet metal require additional, more costly, processes (secondary operations).

Figure 69: Counterbore



Note:

Coined counterbores are not recommended for stainless steel .

2.4.20 COUNTERBORE - COINED

GUIDELINE

RULE:

1. A coined counterbore is a stepped hole with a maximum counterbore depth equal to 50% of the material thickness. See Figure 70, Counterbore - Coined.
2. The maximum diameter of the counterbore to the through-hole is based upon the material criteria (for example, hardness, thickness). See a Producibility Engineer for specific design recommendations.

NOTE

A coined counterbore usually results in surface bulge. See Figure 70, Counterbore - Coined.

Coined counterbores are not recommended for stainless steel.

REASON: Coining a counterbore in sheet metal deeper than the rule specifies may cause excessive material bulging or fracturing and hole distortion leading to manufacturing and/or quality problems.

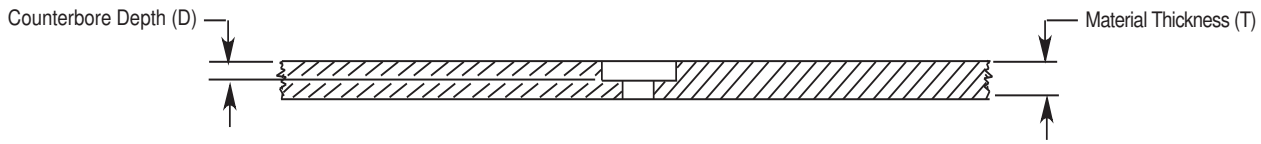
ALTERNATIVES: There are other methods of producing a counterbore in sheet metal. Examples include, but are not limited to:

- Electrical-discharge machining (EDM)
- Metal cutting

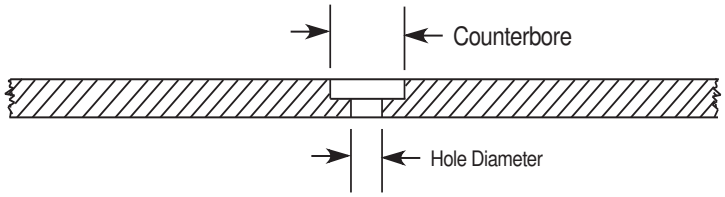
If the design is not impaired by counterbore material bulging a note can be added to the drawing/database stating material bulging is allowable (be specific on location).

ALTERNATIVE IMPACT: These other methods of producing counterbores in sheet metal require special tooling and/or secondary operations that lead to increased process and tooling costs.

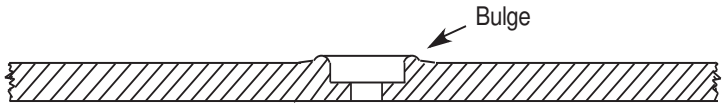
Figure 70: Counterbore - Coined



Rule: D (maximum) = 50 % T



Rule: Ratio between hole diameter and counterbore have no specific limitations. See Producibility Engineer.



Note: Coined counterbore may create bulge.
Coined counterbores are not recommended for stainless steel.

2.4.21 COUNTERBORE TO COUNTERBORE

PRINCIPLE

RULE:

The minimum recommended distance between the major diameter of two counterbores is eight times the material thickness. See Figure 71, Counterbore to Counterbore.

NOTE

Coined counterbores are not recommended for stainless steel.

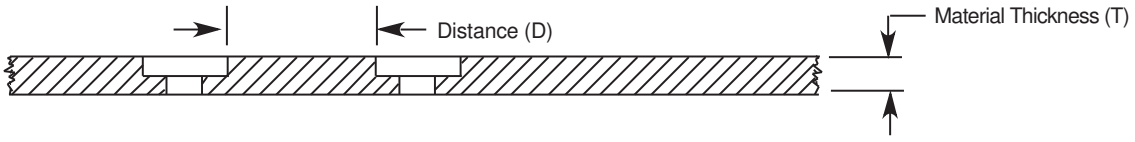
REASON: Counterbores coined too close together may result in distortion of the counterbores and/or web area between them.

ALTERNATIVES: There are other methods of producing counterbores in sheet metal closer than the rule suggests. Examples include, but are not limited to:

- Specially designed tooling/dies
- Metal cutting

ALTERNATIVE IMPACT: These other methods of producing a counterbore in sheet metal require special tooling and/or secondary operations that lead to increased process and tooling costs.

Figure 71: Counterbore to Counterbore



Rule: D (minimum) = 8 T

Note:

Coined counterbores are not recommended for stainless steel .

2.4.22 COUNTERBORE TO EDGE

PRINCIPLE

RULE:

The minimum recommended distance from the major diameter of the counterbore to the edge of the material is four times the material thickness. See Figure 72, Counterbore to Edge.

NOTE

Coined counterbores are not recommended for stainless steel.

REASON: Counterbores formed too close to an edge may result in distortion of the counterbore or web area between the counterbore and the edge, creating a bulge.

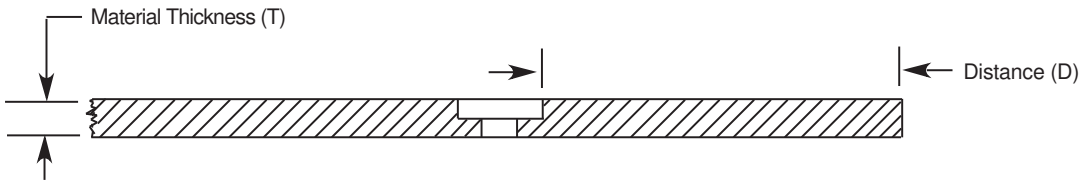
ALTERNATIVES: Extend the edge local to the counterbore by adding a tab to satisfy the rule.

There are other methods of producing counterbores closer to an edge than the rule suggests. Examples include, but are not limited to:

- Specially designed tooling/dies
- Machining

ALTERNATIVE IMPACT: These other methods of producing counterbores in sheet metal require special tooling or secondary operations that lead to increased process and tooling costs.

Figure 72: Counterbore to Edge



Rule: D (minimum) = 4 T

Note:
Coined counterbores are not recommended for stainless steel .

2.4.23 COUNTERBORE TO BEND

PRINCIPLE

RULE:

The minimum recommended distance from the major diameter of the counterbore to the inside of the bend is four times the material thickness plus the bend radius. (See Figure 73, Counterbore to Bend).

NOTE

Coined counterbores are not recommended for stainless steel.

REASON: Counterbores positioned too close to a bend may result in distortion of the counterbore.

ALTERNATIVES: Counterbores may be located much closer to a bend than the rule suggests by using relief cuts/slots, effectively adding a tab.

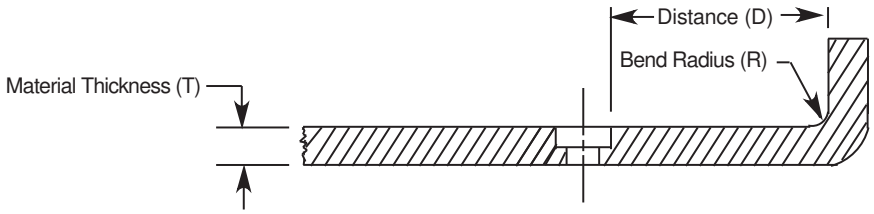
There are other methods of producing counterbores closer to a bend than the rule suggests. Examples include, but are not limited to:

- Specially designed tooling/dies
- Machining

ALTERNATIVE IMPACT: Design considerations or constraints may impair the adding of bend reliefs. They will also make it necessary to incur (minimal) operational or tooling cost increases.

These other methods of producing counterbores in sheet metal require special tooling or secondary operations leading to increased process and tooling costs.

Figure 73: Counterbore to Bend



Rule: D (maximum) = 4 T + R

Note:
Coined counterbores are not recommended for stainless steel .

2.4.24 SEMI-PIERCED HOLE

GUIDELINE

RULES:

1. A semi-pierced hole is a blind hole that is punched to a maximum depth (slug height) of one-half the material thickness. See Figure 74, Semi-Pierced Hole.
2. A blind hole which has been punched and forced back into the material is called a knockout (for example, a removable slug). See a Producibility Engineer for specific design recommendations.

NOTE

A semi-pierced hole is typically used to locate, multiple parts. It may also be produced with or without a web (for example, a removable slug). See Figure 74, Semi-Pierced Hole.

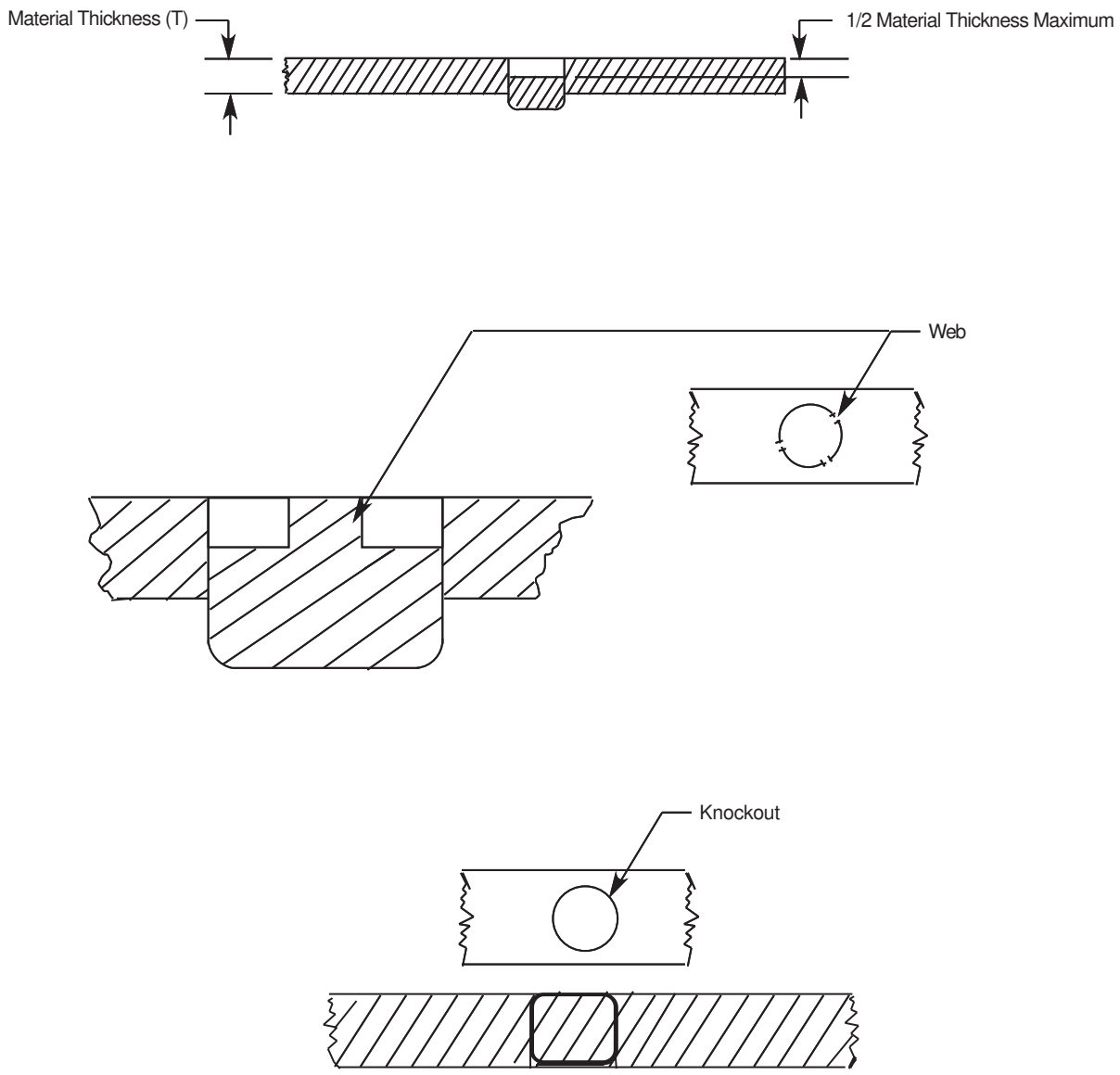
REASO: Material may be pierced to a controlled depth.

ALTERNATIVES: There are other methods of simulating a semi-pierced hole feature in sheet metal and to a depth (slug height) greater than that recommended in the rule. Examples include, but are not limited to:

- Pressed in fasteners
- Use of multiple parts

ALTERNATIVE IMPACT: Positive or negative impacts of alternatives have to be evaluated based upon specific design requirements. See a Producibility Engineer.

Figure 74: Semi-Pierced Hole



2.4.25 SEMI-PIERCED HOLE TO FORM

GUIDELINE

RULE:

The minimum recommended distance from the edge of a semi-pierced hole to a form is three times the material thickness plus the form radius (if applicable). See Figure 75, Semi-Pierced Hole to Form.

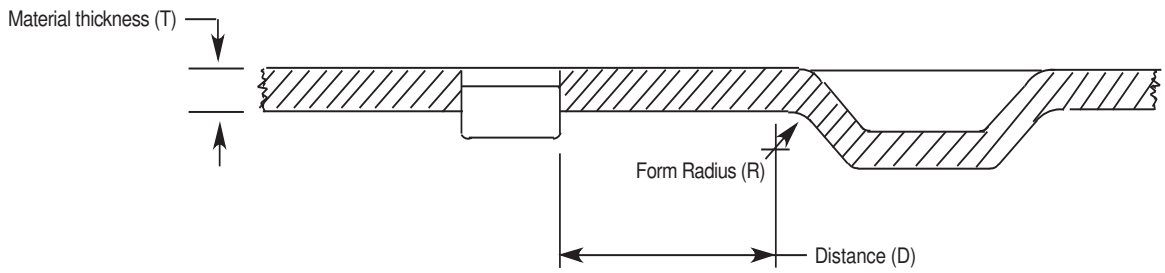
REASON: Semi-pierced holes positioned too close to a formed feature may result in hole distortion leading to manufacturability or quality problems. See Figure 75, Semi-Pierced Hole to Form.

ALTERNATIVES: Semi-pierced holes may be located much closer to a formed feature than the rule suggests by using relief cuts/slots.

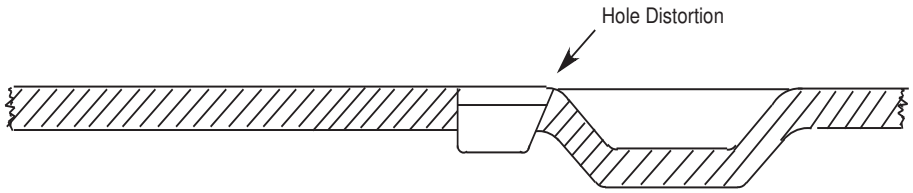
If the design is not impaired by hole distortion a note can be added to the drawing/database stating that hole distortion is allowable.

ALTERNATIVE IMPACT: Design considerations or constraints may impair the adding of form reliefs. They will also make it necessary to incur (minimal) operational or tooling cost increases.

Figure 75: Semi-Pierced Hole to Form



Rule: D (minimum) = 3 T + R



2.4.26 SEMI-PIERCED HOLE TO BEND

GUIDELINE

RULE:

The minimum recommended distance from the edge of a semi-pierced hole to a bend is two times the material thickness plus the bend radius. See Figure 76, Semi-Pierced Hole to Bend.

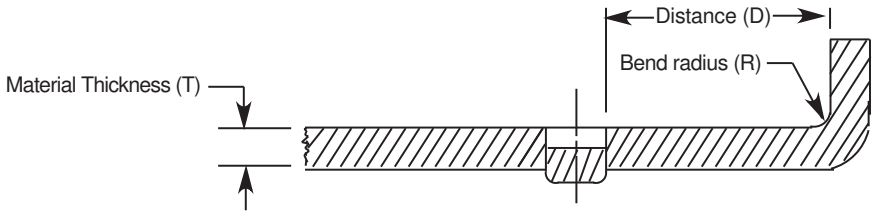
REASON: Semi-pierced holes positioned too close to a bend may result in hole distortion leading to manufacturability and/or quality problems. See Figure 76, Semi-Pierced Hole to Bend.

ALTERNATIVES: Semi-pierced holes may also be located much closer to a bend than the rule suggests by using relief cuts/slots.

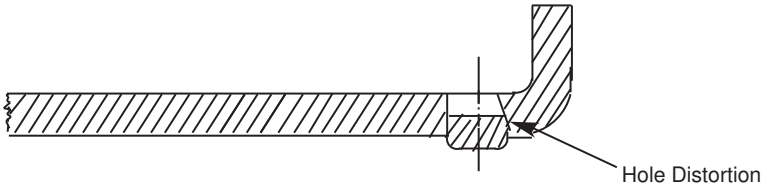
If the design is not impaired by hole distortion a note can be added to the drawing/database stating that hole distortion is allowable (be specific on location).

ALTERNATIVE IMPACT: Design considerations and/or constraints may impair the adding of bend reliefs. They will also make it necessary to incur (minimal) operational and/or tooling cost increases.

Figure 76: Semi-Pierced Hole to Bend



Rule: D (maximum) = 2 T + R



2.4.27 SEMI-PIERCED HOLE TO SEMI-PIERCED HOLE

GUIDELINE

RULE:

The minimum distance between semi-pierced holes is equal to eight times the material thickness. See Figure 77, Semi-Pierced Hole to Semi-Pierced Hole.

REASON: Semi-pierced holes punched too close together may result in warpage of the part and distortion of the holes or the web area between them.

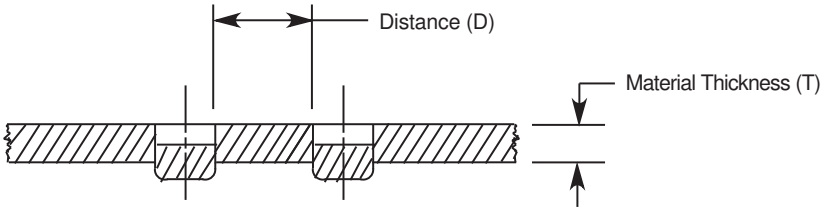
ALTERNATIVES: Combine the semi-pierced holes into a single (larger) semi-pierced hole. Examples include, but are not limited to:

- Two round holes into one slot
- Two slots into one larger cutout

There are other methods of producing a semi-pierced feature on a part, such as using multiple parts (for example, purchased hardware) and assembly techniques (for example, welding).

ALTERNATIVE IMPACT: Positive or negative impacts of the alternatives have to be evaluated based upon specific design requirements. See a Producibility Engineer.

Figure 77: Semi-Pierced Hole to Semi-Pierced Hole



Rule: D (minimum) = 8 T

2.4.28 SEMI-PIERCED HOLE TO EDGE

GUIDELINE

RULE:

A minimum distance must be maintained from a semi-pierced hole to the edge of the material. Semi-pierced holes are not limited to a particular shape or size making it difficult to define. For specific design recommendations, see a Producibility Engineer.

REASON: Semi-pierced holes punched too close to an edge may result in distortion of the hole or the web area between the hole and the edge (creating a bulge) leading to manufacturing or quality problems.

ALTERNATIVES: Extend the edge local to the hole to satisfy the rule. (For example, add a tab).

If the design is not impaired by the edge distortion (bulge), a note can be added to the drawing/database stating that edge distortion is allowable (be specific on location).

ALTERNATIVE IMPACT: Positive or negative impacts of the alternatives have to be evaluated based upon specific design requirements. See a Producibility Engineer.

2.5 FINISHING

Refer to EL-EN727-01, *Finishing Rules Handbook for Sheet Metal*, for finishing requirements.

2.6 JOINING

2.6.1 Slot and Tab

GUIDELINE

RULE:

Slots and tabs may be used as a method of alignment or assembly of two or more parts. (See Figure 78, Slot and Tab).

The slot width must be greater than the maximum material thickness of the tab. The tab length is recommended to be equal to material(s) thickness.

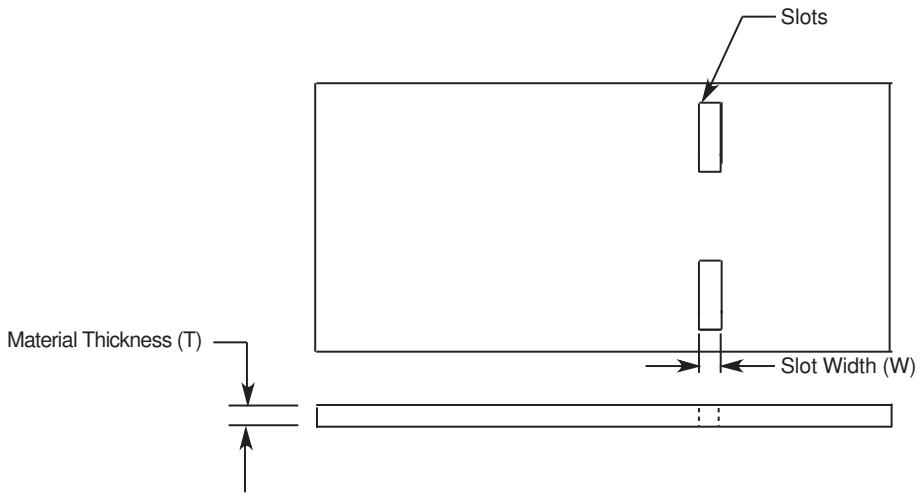
REASON: The rule should be followed to ensure proper mating and assembly of the parts.

ALTERNATIVES: There are other methods of aligning and assembling parts. Examples include, but are not limited to:

- Semi-pierced holes
- Tooling/assembly fixtures
- Welding/rivets/screws

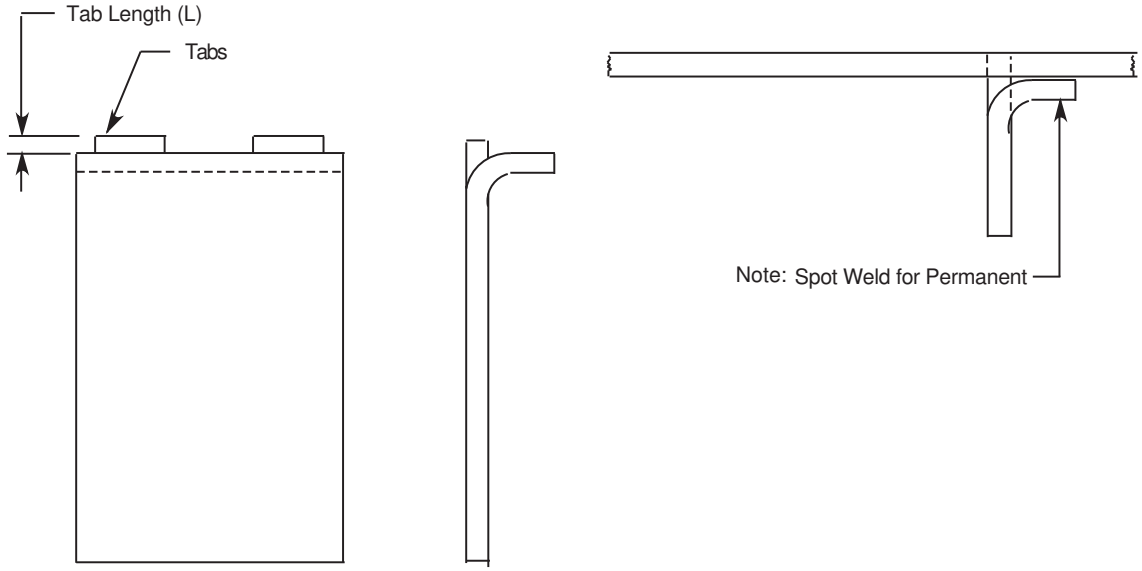
ALTERNATIVE IMPACT: Positive or negative impacts of alternatives have to be evaluated based upon specific design requirements. See a Producibility Engineer.

Figure 78: Slot and Tab



Rule: Slot width W must be greater than material thickness T.
Tab length L is equal to T.

Assembly



2.6.2 RIVETS

INFORMATION

RULE:

Rivets may be used to permanently join two or more parts. See Figure 79, Rivets.

Rivets are classified into the following categories. See a Producibility Engineer for specifications on manufacturing recommendations.

- Solid
- Tubular
- Blind

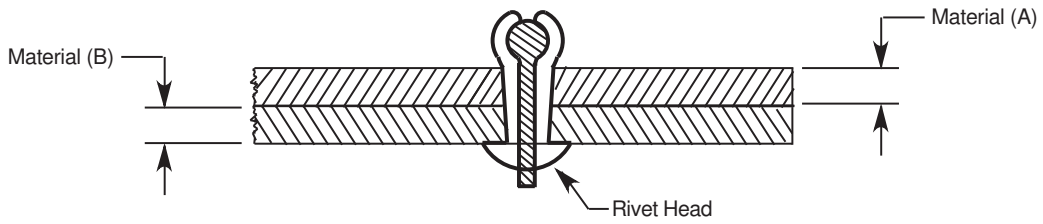
REASON: Rivets are an industry standard and proven method of fastening sheet metal.

ALTERNATIVES: There are other methods of aligning and assembling parts. Examples include, but are not limited to:

- Tooling/assembly fixtures
- Welding/screws/slot and tab

ALTERNATIVE IMPACT: Positive or negative impacts of alternatives have to be evaluated based upon specific design requirements. See a Producibility Engineer.

Figure 79: Rivets



2.6.3 RIVETS - BLIND

GUIDELINE

RULES:

1. When blind rivets are used to align or join two or more parts with round holes, the smaller hole should be located nearest the rivet head (the non-expanding side). See Figure 80, Rivets - Blind.
2. When using blind rivets to align or join two or more parts by means of a slot and a round hole, the slot should be located nearest the rivet head (the non-expanding side). See Figure 80, Rivets - Blind.

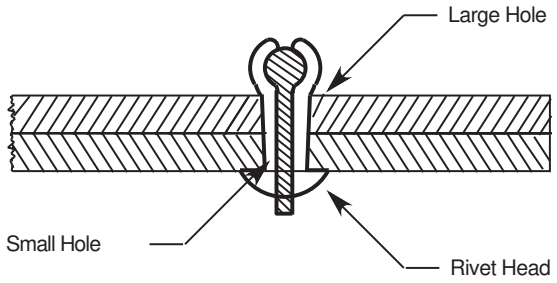
REASON: The rules should be followed to ensure proper seating with respect to the rivets and parts.

ALTERNATIVES: There are other types of rivets and methods of aligning or joining sheet metal parts. Examples include, but are not limited to:

- Solid/tubular rivets
- Welding
- Screws

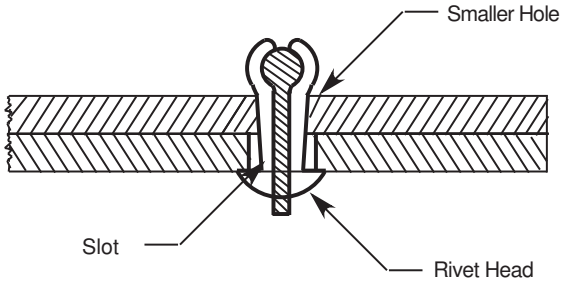
ALTERNATIVE IMPACT: Positive or negative impacts of alternatives have to be evaluated based upon specific design requirements. See a Producibility Engineer.

Figure 80: Rivets - Blind



Rule:

Smaller hole near rivet head.



2.6.4 WELDING

INFORMATION

RULE:

Two typically similar materials may be permanently attached by subjecting them to heat and sometimes pressure, otherwise known as welding. For standard welding symbols, see the following page.

Resistance (spot) welding is the preferred method of welding. Arc welding and gas welding may also be used, but are slower and more expensive processes.

Other welding techniques include, but are not limited to:

- Butt welding
- Projection welding
- Line welding
- Sandwich welding

NOTE

For additional information, refer to *DEC STD 187-0 Mechanical Fabrication Workmanship Standards* and/or ANSI/AWS A2.4 - 79 *American Welding Society Symbols for Welding and Non-destructive testing*.

REASON: Welding is an economical method of joining two or more parts.

ALTERNATIVES: See subhead 2.6 Joining.

ALTERNATIVE IMPACT: Positive or negative impacts of various methods of joining sheet metal have to be evaluated based upon specific design requirements. See a Producibility Engineer.

Figure 81: Standard Welding Symbols

2.6.5 INSERTS

INFORMATION

RULE:

Anything that is permanently put into the material for the purpose of joining or locating is called an insert. Examples include, but are not limited to:

- Quarter-turn screws
- Standoffs
- Press nuts (threaded)
- Studs (threaded)
- Bushings

NOTE

For low carbon steel parts to be plated use low carbon steel inserts.

For aluminum parts use stainless steel inserts for chromate.

For stainless steel parts use stainless steel inserts.

REASON: Inserts are an economical method of providing a feature in sheet metal for joining, aligning or attaching two or more parts.

ALTERNATIVES: See subhead 2.6 Joining.

ALTERNATIVE IMPACT: Positive or negative impacts of various methods of joining sheet metal have to be evaluated based upon specific design requirements. See a Producibility Engineer.

2.6.6 AKH (RIVET, FASTENER)

INFORMATION

RULE:

A headless, self-piercing, auto locking system (AKH), may be used as a method of permanently joining two or more parts. See Figure 82, AKH (Rivet, Fastener).

The AKH process may be used on pre-finished materials. Components of the system come in two finishes, phosphate and zinc, which may be finished along with the mating parts.

Features of the process include:

- Self-piercing
- Headless
- Flush (both sides)
- May be used to laminate nonferrous materials
- May be used for dissimilar materials

NOTE

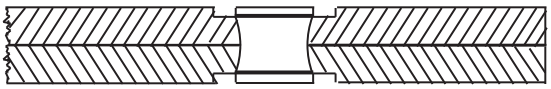
For more detailed information on the process and its capabilities, see a Producibility Engineer.

REASON: This process has been proven to be an effective method of permanent assembly.

ALTERNATIVES: See subhead 2.6 Joining.

ALTERNATIVE IMPACT: Positive or negative impacts of various methods of joining sheet metal have to be evaluated based upon specific design requirements. See a Producibility Engineer.

Figure 82: AKH (Rivet, Fastener)



Note:

For further detailed process information and capabilities, See a Producibility Engineer.

GLOSSARY

artificial intelligence (AI): Intellectual reasoning simulated through the use of computers.

assembly: The performing of specific tasks in predetermined order to create a whole.

bend: The straining of material, usually sheet metal, from a straight form into a curved form.

blanking: To cut or shear a part to a predetermined contour.

bulge: To protrude outward from an edge.

burr: The side of a punched surface that presents a rough edge around its periphery.

chamfer: An oblique face formed at a corner.

chemical etching: The corrosive action or eating away of metal through the use of acids.

coined: To squeeze the metal in a confined or restrained state.

curl: An edge bent into a loop or circle along a straight axis.

depression: A recessed indentation to a surface.

die clearance: The space per side, between the punch and die.

dies: Equipment consisting of a pair, or combination of pairs, of mating members for producing work or parts in presses.

dimple: A conical flange usually encircling a hole.

distortion: Any deviation from a desired shape.

edge: A thin or sharp side in which a surface terminates or ends.

electrical discharge machining (EDM): The process of removing or burning of metal through means of electrical arcing.

embossment: Shallow or raised indentations with little or no change in material thickness.

extrude: To form metal by forcing through a die orifice.

flange: A projecting surface to give additional strength or support.

form: To make a change in the shape of sheet metal which does not reduce the metal thickness.

gusset: A form or piece of material to improve or reinforce a bend.

hard tooling: Equipment or processes associated with high volume manufacturing (for example, dies).

hole: An opening through something, such as sheet metal.

laser: A device that produces a very narrow beam of extremely intense light which can be used in industrial processes.

louvers: Any series of narrow, slanting, formed openings.

low carbon steel: Steel with a carbon content up to and including 0.18% (AISI C1018)

machining: To make or finish by mechanical means through the use of cutting tools.

manufacturability: To be producible on a large scale by means of equipment or machinery.

offset: A part or surface that branches off from the main part or surface.

oil-canning: The flexing or distortion of sheet metal, usually caused by uneven or extensive cutting on a surface.

piercing/punching: The process of die cutting holes in sheet metal.

plate: To coat metal with a thin film of copper, nickel, etc., usually for decorative or shielding applications.

process: The systematic series of operations to achieve some end.

radius: A circular area determined by a straight line extending from the center of a circle to a circumference .

relief: Clearance obtained by the removal of metal, usually associated with bends or forms.

secondary operations: A process which is required after or proceeding the original or primary process.

shear: To cut metal, or other material, by means of two cutting edges.

sheet metal: A piece of metal up to one quarter inch in thickness.

spherical: A round surface on which all points are equidistant from the center.

soft tooling: Equipment or processes usually associated with low volume manufacturing (for example, N.C. Punch Press).

warpage: To become bent or twisted from a straight or flat form.

weld: To join pieces of metal by means of fusion.

APPENDIX A REFERENCED DOCUMENTS

A.1 Digital Documents, EL-Class

Document Number	Document Title
EL-00187-00	*DEC STD 187-0 Mechanical Fabrication Workmanship Standards*
EL-EN727-01	*Finishing Rules Handbook for Sheet Metal*

A.2 Documents External to Digital

Document Number	Document Title
ANSI/AWS A2.4-79	*American Welding Society Symbols for welding & Non-Destructive Testing*
AWS STANDARD A2.4-79	*American Wire Standard*

A.3 Ordering Information

Use VTX SMC to order copies of EL-class Digital documents from Standards and Methods Control. Send distribution questions to JOKUR::SMC or call DTN: 234-4423.

Following are the sources for documents not available from Standards and Methods Control. Copies may also be available from your local Digital library.

- ANSI, IEC, and ISO Documents
  American National Standards Institute (ANSI)
  1430 Broadway, New York, NY 10018
  Telephone (212) 354-3300 x479
- American Society for Testing and Materials (ASTM)
  1916 Race Street
  Philadelphia, PA 19103

INDEX

A

AKH (rivet, fastener), 190

B

bending, 28
bend radius, 30
bend relief, 38
bend to bend, 40
bend to edge, 34
bend to hole, 36

C

corner condition - chamfer, 122
corner condition - radius, 120
corner conditions, 118
counterbore, 160
counterbore - coined, 162
counterbore to bend, 168
counterbore to counterbore, 164
counterbore to edge, 166
countersink, 148
countersink - coined, 152
countersink - formed, 150
countersink to bend, 158
countersink to countersink, 154
countersink to edge, 156
curl, 64
curl to bend, 68
curl to curl, 70
curl to hole, 66

D

diameters, 10
dimple, 44
dimple to bend, 50
dimple to dimple, 52
dimple to edge, 48
dimple to hole, 46

E

edge, 102
edge stamping process, 102
embossment, 98
embossment shapes, 100
extruded, hole, 124
extruded hole - roll form threading, 126
extruded hole to bend, 132
extruded hole to edge, 130
extruded hole to extruded hole, 128

F

finishing, 179
forms, 28

G

gusset, 54
gusset to bend, 62
gusset to edge, 60
gusset to gusset, 58
gusset to hole, 56

H

headless self-piercing, auto locking, 190
HEM, 72
hem - closed, 78
hem - open, 76
hem - teardrop, 74
hem to bend, 84
hem to edge, 82
hem to hem, 86
hem to hole, 80
holes, 8
holes to form, 14
hole through a bend, 26
hole to bend, 16
hole to edge, 22
hole to hole, 20

I

inserts, 188
introduction, 2

J

joining, 180

L

lance - closed, 136
lance - open, 134
lance to bend, 140
lance to edge, 138
lance to hole, 142
lance to lance, 144
louvers, 146

N

notch, 104
notch through a bend, 110
notch to bend, 108
notch to hole, 106
notch to notch, 112

O

offset bends, 42

P

piercing, 8

R

rib, 88
rib to bend, 94
rib to edge, 92
rib to hole, 90
rib to rib, 96
rivets, 182
rivets - blind, 184

S

semi-pierced hole, 170
semi-pierced hole to bend, 174
semi-pierced hole to edge, 178
semi-pierced hole to form, 172
semi-pierced hole to semi-pierced hole, 176
slot, 12
slot and tab, 180
slot to bend, 18
special features, 124
stamping process, edge, 102

T

tab, 114
tab to tab, 116
teardrop, hem, 74

W

welding, 186

```
+---------------------------------------------------------------------+
|                          READER COMMENTS                            |
|                                                                     |
| Your  comments  and suggestions  will help  Standards and Methods   |
| Control improve their services and documents.                       |
+---------------------------------------------------------------------+
```

Did you request this document? _____ If so, did it arrive within a satisfactory period of time? _____ Please comment.

What are your impressions of this document? Consider format, organization, completeness, readability, and illustrations.

————————————**FOLD ON THIS LINE**———————————————————————— -

Did you find technical or clerical errors in this document? If so, please specify the page number(s) and the error(s).

Are the instructions for the update package clear? _____
Was an index available? _____ If not, is one needed? _____
Do you have other suggestions for improving this document?

The following information is optional:

Name _____ Mailstop _____

Department _____ Node _____

Send your comments to JOKUR::PROJECTS, or fold, staple, and send this page through interoffice mail to:

```
+------------------------------+
|       READERS' COMMENTS      |
| STANDARDS AND METHODS CONTROL |
|           NRO5/I2            |
+------------------------------+
```